DOCUMENT RESUME

ED 351 518 CE 062 320

AUTHOR	Walden, Charles; Cole, Phyllis
TITLE	Machine Shop. Instructor Key. Supplementary Units.
INSTITUTION	Missouri Univ., Columbia. Instructional Materials Lab.
SPONS AGENCY	Missouri State Dept. of Elementary and Secondary Education, Jefferson City. Div. of Vocational and Adult Education.
PUB DATE	Jul 91
NOTE	306p.; For related documents, see ED 316 749-756 and ED 336 523.
AVAILABLE FROM	Instructional Materials Laboratory, University of Missouri, 2316 Industrial Drive, Columbia, MO 65202 (order no. 90-9200-S).
PUB TYPE	Guides - Classroom Use - Teaching Guides (For Teacher) (052)
EDRS PRICE	MF01 Plus Postage. PC Not Available from EDRS.
DESCRIPTORS	Behavioral Objectives; Competence; *Competency Based Education; *Course Content; Job Skills; *Learning Disabilities; Machine Tool Operators; *Machine Tools; *Numerical Control; Postsecondary Education; Secondary Education; *Special Needs Students; State Curriculum Guides; Teaching Methods; Vocational Education

ABSTRACT

These supplementary units are designed to help students with special needs learn and apply machine shop skills. Nine competencies that are difficult for special needs students to grasp or that would help them get a future job in the field were chosen from the regular machine shop curriculum. Specific objectives for these competencies are listed at the beginning of each of the nine self-paced units. The units contain large print, simplified text, and numerous drawings to aid in comprehension. Each unit contains the following sections: introduction, objectives, study questions, and student assessment; terms and equipment list; information sheets for each competency; skill sheets (exercises); activity sheets; and unit tests and performance checklists. A section on instruction information contains notes to the instructor, suggested resources, and answer Keys. The nine units cover the following topics: calculating speeds and feeds, using a rule, micrometers, blades for sawing, selecting and sharpening a drill, using a computer numerically controlled (CNC) machine, CNC machine safety, tool variables for machining, and grinding the lathe toolbit. (KC)

****
* Reproductions supplied by EDRS are the best that can be made *
* from the original document. *
****

M091CD05

90-9200-K

ED351518

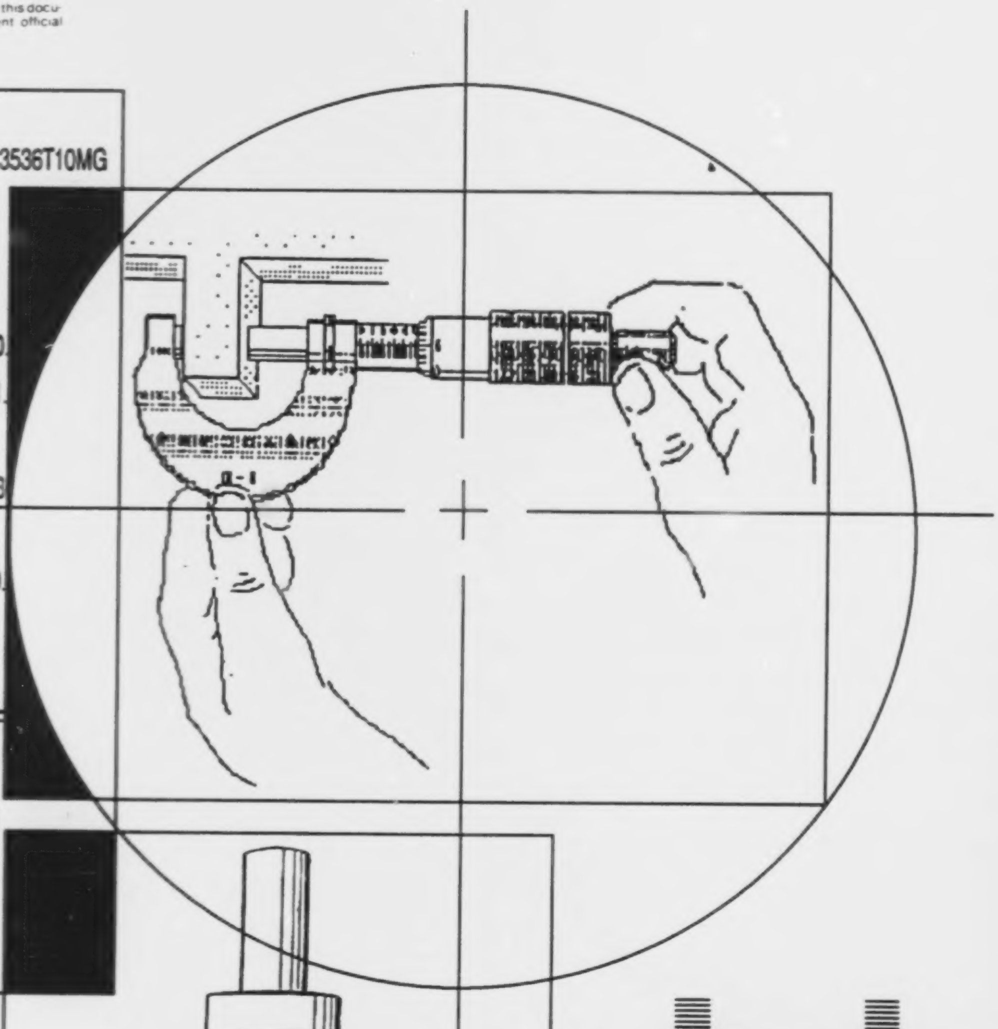
CE062320

Supplementary Units

MACHINE SHOP INSTRUCTOR KEY

U.S. DEPARTMENT OF EDUCATION
Office of Educational Research and Improvement
EDUCATIONAL RESOURCES INFORMATION CENTER (ERIC)
☑ This document has been reproduced as received from the person or organization originating it
☐ Minor changes have been made to improve reproduction quality
- Points of view or opinions stated in this document do not necessarily represent official OERI position or policy

```
%
N1691G00G75X0.3536Y0.3536T10MG
N2X0.0Y0.0Z0.0
N3Z0.05
N4G01Z-0.05F10.0
N5G01X0.7071Y0.7071F20.
N6G03Y-2.1214I-1.0607J-1.
N7G01X-0.7071Y0.7071
N8G02Y0.7072I-0.3536J0.3
N9G00
N10G00X1.6464Y1.1464Z0.
N11Z0.05
N12G01Z-0.05F10.0
N13G03X-0.5Y0.5I-0.5J0.0F
N14G01X-3.0
N15G03X-0.5Y-0.5I0.0J-0.5
N20G01Y3.0
N21G00
N22G90G00X-2.0y-1.5M2
```



IMI

"PERMISSION TO REPRODUCE THIS MATERIAL IN OTHER THAN PAPER COPY HAS BEEN GRANTED BY

H Schlichting

TO THE EDUCATIONAL RESOURCES INFORMATION CENTER (ERIC)."

BEST COPY AVAILABLE

SUPPLEMENTARY UNITS

MACHINE SHOP

Written by:

Charles Walden

Phyllis Cole

Edited by:

Betty Gibb

University of Missouri-Columbia
Columbia, Missouri

July 1991

SUPPLEMENTARY UNITS, MACHINE SHOP

TABLE OF CONTENTS

Preface v
Acknowledgements vi
General Notes to the Instructor vii
References and Materials viii
Competencies ix
Class Progress Chart xi
Follow-up Questionnaire xiii
Answer Keys xv
Calculate Speeds and Feeds . . . . . Unit 1
Using a Rule . . . . . Unit 2
Micrometers . . . . . Unit 3
Blades for Sawing . . . . . Unit 4
Selecting and Sharpening a Drill . . . . . Unit 5
Using a CNC Machine . . . . . Unit 6
CNC Machine Safety . . . . . Unit 7
Tool Variables for Machining . . . . . Unit 8
Grinding the Lathe Toolbit . . . . . Unit 9

Copyright 1991

The Curators of the University of Missouri

All Rights Reserved

Produced by the Instructional Materials Laboratory
University of Missouri-Columbia, Columbia, Missouri 65211

July 1991
Catalog Number 90-9200-S

This activity was supported in whole or in part by funds from the Department of Elementary and Secondary Education, Division of Vocational and Adult Education. However, the opinions expressed herein do not necessarily reflect the position or policies of the Missouri Department of Elementary and Secondary Education or the Division of Vocational and Adult Education, and no official endorsement should be inferred.

SUPPLEMENTARY UNITS, MACHINE SHOP

PREFACE

These supplementary units are designed to help students with special needs learn and apply machine shop skills. Nine competencies that are difficult for special needs students to grasp or that would help them get a future job in the field were chosen from the regular Machine Shop curriculum.

Specific objectives for these competencies are listed at the beginning of each of the nine self-paced units. Skill sheets, activity sheets and unit tests reinforce and assess the student's growing knowledge. The units contain large print, simplified text and numerous drawings to aid in comprehension.

Each unit within this curriculum is color coded to represent a separate section. Color coding is explained as follows:

White:	Introduction, Objectives, Study Questions and Student Assessment. These pages present unit and specific objectives the student must meet to complete the unit successfully. The introduction contains a brief motivational statement describing the purpose and content of the unit. Student Assessment allows the student to determine whether to study the material in the unit or move directly to the unit test.
Salmon:	Terms and equipment. This section introduces new terms necessary for the student to understand and answer the study questions shown in the unit. Equipment is also introduced and illustrated.
Canary:	Information. These pages present detailed information and explanations for each competency.
Green:	Skill Sheets. These pages provide paper-and-pencil exercises to develop and reinforce the skills presented in the unit.
Blue:	Activity Sheets. Various activities are presented to increase student involvement and provide extra practice in performing the procedures.
Pink:	Unit Test and Performance Checklist. These pages are used by the instructor to evaluate student performance. Different forms of review, including identification, matching, multiple-choice and short-answer exercises are provided.

"Instructor information," located at the front of the book, contains notes to the instructor, helpful resources and answer keys.

SUPPLEMENTARY UNITS, MACHINE SHOP

ACKNOWLEDGEMENTS

These supplementary units for Machine Shop represent the combined efforts of the following individuals:

Writers:

Charles Walden, Manager Engineering Lab, Mechanical & Aerospace Engineering, University of Missouri Columbia
Phyllis Cole, Vocational Resource Educator, N. S. Hillyard AVTS, St. Joseph

Advisory Committee:

Pete Lopez, Industrial Education, N. S. Hillyard AVTS, St. Joseph
Phyllis Cole, Vocational Resource Educator, N. S. Hillyard AVTS, St. Joseph
Bill Kamp, Industrial Education, Laclede AVTS, Lebanon
Brenda Evans, Vocational Resource Education, Laclede AVTS, Lebanon
Kirt Salmons, Industrial Education, State Fair Community College, Sedalia
Gay Groves, Vocational Resource Educator, State Fair Community College, Sedalia
Mel Gage, Industrial Education, Linn Technical College, Linn

Department of Elementary and Secondary Education:

Bob Larivee, Director, Vocational Special Needs and Guidance Services
Dick Omer, Assistant Director, Vocational Special Needs Education
Iva Presberry, Supervisor, Vocational Special Needs Education
Delbert Lund, Supervisor, Vocational Planning and Evaluation

Instructional Materials Laboratory:

Harley Schlichting, Director
Amon Herd, Associate Director
Phyllis Miller, Assistant Director
Dan Stapleton, Assistant Director
Betty Gibb, Assistant Editor
Fiona Gammonley, Graphic artist

Content Review and Evaluation:

Charles Walden, Manager Engineering Lab, Mechanical & Aerospace Engineering, University of Missouri Columbia

Fieldtesting and Evaluation:

Bill Kamp, Industrial Education, Laclede AVTS, Lebanon
Jack Miller, Industrial Education, Clinton AVTS, Clinton
Ralph Johns, Industrial Education, New Madrid AVTS, New Madrid
LeAnn Campbell, Vocational Resource Educator, Lamar AVTS, Lamar
Dana Robinson, Industrial Education, Lamar AVTS, Lamar
Annabel Gooch, Vocational Resource Educator, Davis H. Hart Mexico Area School, Mexico
Raona Hentz, Vocational Resource Educator, Poplar Bluff AVTS, Poplar Bluff
Bill Daniel, Industrial Education, Hannibal AVTS, Hannibal
James Jemes, Industrial Education, Four Rivers AVTS, Washington
Pat Cleveland, Industrial Resource Educator, Clinton

GENERAL NOTES TO THE INSTRUCTOR

Step 1: Read through the information in each unit before presenting it to the student.

Step 2: Make sure all equipment to be used in each unit is available.

Step 3: Provide student with the Introduction pages.

Step 4: Discuss the Introduction and Objectives on the Introduction pages.

Step 5: Have the student complete the "Are You Ready?" section.

Step 6: If the student elects to complete the Unit Test and Performance Checklist and does so to your satisfaction, have him or her proceed to the next unit.

Step 7: Discuss the special terms.

Step 8: Provide the student with the Information section.

Step 9: Select a means to present the Information section.

A. Have the student study the information independently.
B. Go through the information as a group.
C. Go through the material as a group; then have students work independently.

Step 10: Give the student time to learn the information. (You, the instructor, decide how much time the student needs.)

Step 11: Have the student complete all the Skill Sheets. Skill Sheets reinforce skills necessary to understand and apply the information in the unit. They may also be used by the student as a self-evaluation tool or by the class as a discussion aid.

Step 12: Complete Activity Sheets.

Step 13: When the student is ready, use the Unit Test and Performance Checklist to evaluate the student's performance for each unit.

REFERENCES

1. Olivo, C. Thomas. Machine Tool Technology and Manufacturing Processes. C. Thomas Olivo Associates. Albany, NY: 1987.
2. Walker, John R. Modern Metalworking. The Goodheart-Willcox Company, Inc. South Holland, IL: 1985.

SUPPLEMENTARY UNITS, MACHINE SHOP

COMPETENCIES

Competencies developed in these supplementary units are based on selected areas of the Machine Shop Competency Profile developed in 1988.

Rating Scale:	3	Mastered--can work independently with no supervision
	2	Requires Supervision--can perform job completely with limited supervision
	1	Not Mastered--requires instruction and close supervision
	N	No Exposure--no experience or knowledge in this area

B8 **Calculate feeds and speeds** (Unit 1)

C3 **Measure workpiece with pocket rule** (Unit 2)

C11 **Measure workpiece with micrometers** (Unit 3)

G3 **Select proper blade type for sawing operations and materials** (Unit 4)

H4 **Select proper drill type based on job requirements** (Unit 5)

H18 **Sharpen drills with grinding attachments and/or specialized grinders** (Unit 5)

I4 **Determine cutting tool variables prior to use** (Unit 6)

I6 **Free-hand grind turning and facing tools** (Unit 7)

M1 **Employ CNC machine safety guidelines consistently** (Unit 8)

M6 **Machine workpiece with CNC machine** (Unit 9)

Supplementary Units, Machine Shop

Class Progress Chart

										STUDENT NAME
										WHAT THE STUDENT CAN DO
										Calculate feeds and speeds
										Measure workpiece with a pocket rule
										Measure workpiece with micrometers
										Select proper blade type for sawing operations and materials
										Select and sharpen drill
										Determine cutting tool variables prior to use
										Free-hand grind turning and facing tools
										Employ CNC machine safety guidelines consistently
										Machine workpiece with CNC machine

Special Needs Follow-Up Questionnaire

Staff members, writers and advisors have worked to make these instructional materials easy--both to use and to read. We welcome your suggestions and/or corrections. Please return this questionnaire with your comments to:

Special Needs Editor
Instructional Materials Laboratory
8 London Hall
University of Missouri-Columbia
Columbia, Missouri 65211

Thank you for your time and consideration.

Supplementary Units for

Machine Shop

My overall rating of this publication is:

Excellent Very Good Good Fair Poor

To improve the materials, IML should:

Other Comments:

Optional:

Name:

Address:

CALCULATE FEEDS AND SPEEDS
ANSWER KEY

SKILL SHEET 1

1. Spindle speed
2. Feed
3. RPM
4. Cutting speed
5. Increased
6. Decreased
7. Increased
8. Decreased
9. Decreased
10. Increased

SKILL SHEET 2

1. A. $\frac{\text{Cutting speed X 12}}{\text{Drill diameter X pi (3.1416)}}$

   B. $\frac{\text{Cutting speed X 1000}}{\text{Drill diameter X pi (3.1416)}}$

2. Cutting speed = 200 FPM

   $\frac{200 \text{ X } 12}{.75 \text{ X } 3} = \frac{200 \text{ X } 4}{.75} = \frac{800}{.75} = 1067 \text{ RPM}$

4. Cutting speed = 76.2

   $\frac{76.2 \text{ X } 1000}{8 \text{ X } 3.1416} = \frac{76200}{15.12} = 3033 \text{ RPM}$

SKILL SHEET 3

1. $\frac{\text{Cutting speed X 12}}{\text{Work diameter X pi (3.1416)}}$

2. $\frac{\text{Cutting speed X 1000}}{\text{Work diameter X pi (3.1416)}}$

3. Cutting speed = 300 FPM

   $\frac{300 \text{ X } 12}{5 \text{ X } 3} = \frac{300 \text{ X } 4}{5} = \frac{1200}{5} = 240 \text{ RPM}$

4. Cutting speed = 70

   $\frac{70 \text{ X } 1000}{13 \text{ X } 3.1416} = \frac{7000}{40.82} = 1715 \text{ RPM}$

SKILL SHEET 4

1. When using a CNC machine or one with power feed.

2. ipm (inches per minute) or meters/min

3. Spindle speed (RPM)
   Feed per tooth (R)
   Number of teeth (T)

4. F = RPM X R X T

   Cutting Speed = 400

   $\frac{400 \text{ X } 12}{.5 \text{ X } 3} = \frac{400 \text{ X } 4}{.5} = \frac{1600}{.5} = 3200 \text{ RPM}$

   RPM = 3200
   R = .010
   T = 2
   F = 3200 X .010 X 2
   F = 64 ipm

XVI.

UNIT TEST

1. B
2. A
3. D
4. C
5. 1
6. 3
7. 4
8. 2
9. Apply cutting fluids
10. Drill diameter and workpiece material (cutting speed)

11. $\dfrac{300 \times 12}{.75 \times 3} = \dfrac{300 \times 4}{.75} =$ 1600 RPM

12. $\dfrac{100 \times 12}{3 \times 3} = \dfrac{100 \times 4}{3} = \dfrac{400}{3} =$ 133 RPM

13. $\dfrac{15 \times 1000}{30 \times 3} = \dfrac{15000}{90} =$ 167 RPM

14. Feed per tooth, RPM, Number of teeth

15. F = R X T X RPM
    R = 0.15
    T = 2
    RPM =
    $\dfrac{200 \times 12}{.25 \times 3} = \dfrac{200 \times 4}{.25} = \dfrac{800}{.25} =$ 3200

    F = 0.25 X 2 X 3200 = 1.600

15

USING A RULE

ANSWER KEY

SKILL SHEET 1

1. 3"
2. 1"
3. 5"

SKILL SHEET 2

1. 1 1/2"
2. 4 1/2"
3. 2 1/2"
4. 5 1/2"

SKILL SHEET 3

1. 3 1/4"
2. 4 1/2"
3. 1 1/4"
4. 5 3/4"
5. 2 1/2"

SKILL SHEET 4

1. 1 1/8"
2. 3 5/8"
3. 1 1/2"
4. 2 7/8
5. 5 3/4

SKILL SHEET 5

1. 9/16"
2. 2 5/16"
3. 13/16"

SKILL SHEET 6

1. 31/32"
2. 5/8"
3. 1 5/32"
4. 1 3/4"

SKILL SHEET 7

1. 19/64"
2. 41/64"
3. 47/64"
4. 7/8"
5. 1 1/4"
6. 2 5/64"

16

MICROMETERS
ANSWER KEY

SKILL SHEET 1

1. DO
2. DON'T
3. DON'T
4. DO
5. DON'T
6. DON'T
7. DO
8. DO
9. DO
10. DO

SKILL SHEET 2

1. ANVIL
2. SPINDLE
3. LOCK
4. SLEEVE
5. RACHET STOP
6. THIMBLE
7. FRAME
8. SLEEVE
9. THIMBLE
10. SPINDLE AND SCREW

SKILL SHEET 3

A. .000
B. .025
C. .100
D. .200
E. .325
F. .450
G. .525
H. .775
I. 1.000

ACTIVITY SHEET 1 & 2 - Instructor's discretion

UNIT TEST

1. .325
2. .175
3. .500
4. .067
5. .033
6. .625
7. .103
8. .022
9. .292
10. .435
11. N
12. G
13. M
14. F
15. L
16. E
17. K
18. D
19. J
20. C
21. B
22. I
23. H
24. A

SELECTING PROPER DRILL

ANSWER KEY

SKILL SHEET 1

1. E
2. C
3. B
4. A
5. D

SKILL SHEET 2

1. Body
2. Shank
3. Tang
4. Margin
5. Flute
6. Lip or cutting edge
7. Two-flute twist drill
8. Oil-hole drill
9. Straight-flute drill

SKILL SHEET 3

1. #80
2. 1/64
3. F
4. 19/32
5. K
6. #69
7. Q
8. #53
9. 15/64
10. 5/8

ACTIVITY SHEETS

Instructor's discretion

UNIT TEST

1. Body
2. Shank
3. Tang
4. Lip
5. Flute
6. Margin
7. E
8. C
9. D
10. B
11. A
12. A, D, F
13. C
14. B
15. B

BLADES FOR SAWING

ANSWER KEY

SKILL SHEET 1

1. skip
2. standard
3. hook
4. tooth spacing
5. tooth back
6. rake angle
7. tooth gullet
8. tooth face
9. C
10. B
11. A
12. D

SKILL SHEET 2

1. A
2. C
3. A & D
4. B

ACTIVITY SHEETS 1 & 2

Instructor's discretion

UNIT TEST

1. D
2. C
3. B
4. A
5. D
6. A
7. C
8. B
9. A, B, E
10. B
11. C

13. A & D

USING A CNC MACHINE

ANSWER KEY

SKILL SHEET 1

1. Z
2. X
3. Y
4. X
5. Y
6. Z
7. Milling machine
8. Control Panel
9. Electrical cabinet
10. Servo motor
11. Any two
    - Repeatability
    - Accuracy
    - Higher productivity
    - More flexibility
12. Origin
13. plus (+) and minus (-)
14. Incremental
15. Computer disk or punched paper tape

SKILL SHEET 2

OPERATIONS

hole drilling
slot milling

Feedrate Formula

F = R X T X RPM
R = Feed per tooth
T = Number of teeth

Spindle speed (RPM) formula

$$\text{RPM} = \frac{\text{Cutting speed X 4}}{\text{Cutter dia.}}$$

RPM = 6400
F = 128 ipm

SKILL SHEET 3

1. C
2. D
3. E
4. A
5. B
6. F

Part 2.

M03	Spindle on
M08	Coolant on
G00	Rapid travel
G01	Linear movement
M05	Spindle off
M06	Tool change
M09	Coolant off
M02	End program

SKILL SHEET 4

1. Manuscript
2. The vertical distance the tool is set above the workpiece when making non-cutting motions
3. B
4. C
5. A
6. D
7. Because an untried program may contain errors that will cause accidents or damage to tooling
8. Single-step

UNIT TEST

1. B
2. H
3. E
4. A
5. D
6. C
7. G
8. F
9. Any two
    - Repeatability
    - Accuracy
    - Productivity
    - Flexibility
10. C
11. A
12. D
13. B
14. E
15. C

CNC MACHINE SAFETY

ANSWER KEY

SKILL SHEET 1

1. E
2. G
3. H
4. B
5. A
6. I
7. D
8. C
9. F

SKILL SHEET 2

1. Machine controller
2. Machine controller
3. Emergency stop button
4. (Any four in any order)
    1. Do not operate controls unless you have been instructed in their use. Remember, The cycle stop button stops the machine as soon as the command has been performed. The emergency stop button halts the feed of all axes immediately.
    2. Never assume. Ask about anything you are unfamiliar with.
    3. Do not attempt to program a machine without programming manuals.
    4. Maintain a safe space above the workpiece.
    5. Keep your hands away from the rotating spindle and moving turrets.
    6. Know where the tool is going AT ALL TIMES.

UNIT TEST

1. T
2. F
3. T
4. F
5. T
6. F
7. T
8. F
9. T
10. T
11. F
12. D
13. E
14. A
15. F
16. B
17. C

TOOL VARIABLES FOR MACHINING

ANSWER KEY

SKILL SHEET 1

1. B
2. A
3. A

SKILL SHEET 2

1. D
2. A

X mineral oil
X synthetic
X emulsifiable

UNIT TEST

1. C
2. B
3. D
4. A
5. C
6. B
7. A

GRINDING A LATHE TOOLBIT

ANSWER KEY

SKILL SHEET 1

1. A
2. F
3. C
4. G
5. B
6. I
7. D
8. H
9. E
10. RIGHT-HAND TOOL

SKILL SHEET 2

1. Aluminum oxide
2. 1/16-inch
3. To keep the toolbit cool while grinding

SKILL SHEET 3

1. 5 degrees
2. 6 degrees
3. 10 degrees
4. 8 degrees
5. 10 degrees
6. 10 degrees

UNIT TEST

Part 1.

1. A
2. F
3. C
4. G
5. B
6. I
7. D
8. H
9. E

Part 2.

10. C
11. F
12. B
13. H
14. A
15. I
16. J
17. D
18. E
19. G

Part 3.

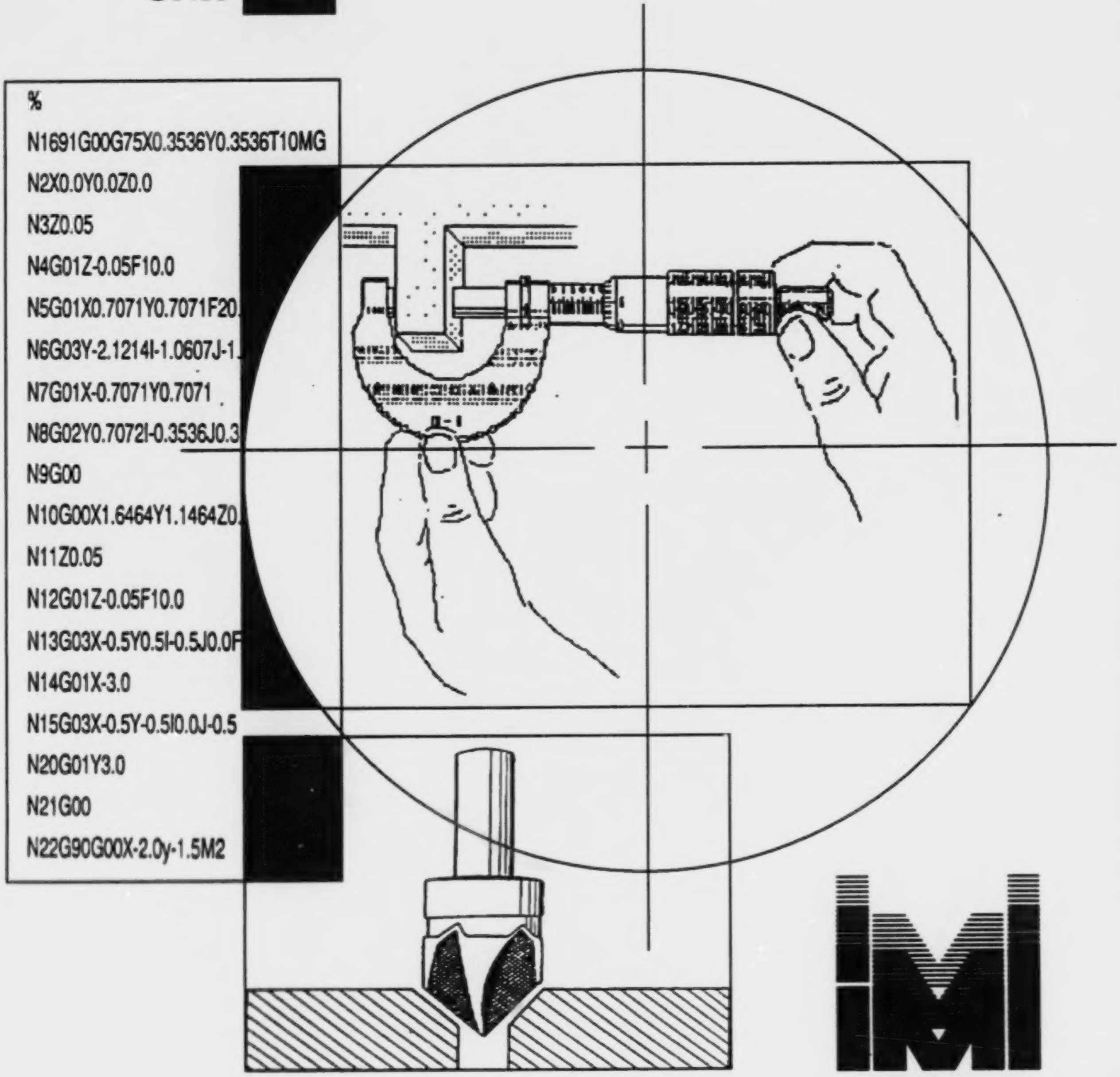
20. side cutting angle
21. side relief angle
22. front cutting angle
23. end relief angle
24. back rake angle
25. side rake angle

Machine Shop

90-9200-S
SUPPLEMENTARY UNITS

CALCULATE SPEEDS AND FEEDS

UNIT 1



```
%
N1691G00G75X0.3536Y0.3536T10MG
N2X0.0Y0.0Z0.0
N3Z0.05
N4G01Z-0.05F10.0
N5G01X0.7071Y0.7071F20
N6G03Y-2.1214I-1.0607J-1
N7G01X-0.7071Y0.7071
N8G02Y0.7072I-0.3536J0.3
N9G00
N10G00X1.6464Y1.1464Z0
N11Z0.05
N12G01Z-0.05F10.0
N13G03X-0.5Y0.5I-0.5J0.0F
N14G01X-3.0
N15G03X-0.5Y-0.5I0.0J-0.5
N20G01Y3.0
N21G00
N22G90G00X-2.0y-1.5M2
```

IMI

CALCULATE SPEEDS AND FEEDS

INTRODUCTION

It's important when you machine any part to run the tool at the proper speed over the workpiece. Proper tool speed will allow you to remove the most material in the least time and produce the best finish possible on the workpiece. Using correct speeds also helps the cutting tool last longer.

UNIT OBJECTIVE

After completing this unit, you should be able to calculate the proper speeds (rpm) and feeds (ipm or mpm) for drilling, turning (lathework) and milling operations.

SPECIFIC OBJECTIVES

After completing this unit, you should be able to:

1. Identify factors that determine speeds and feeds.
2. Calculate the proper rpm setting for drilling on a drill press.
3. Calculate the proper rpm setting for turning a workpiece in a lathe.

4. Calculate the proper rpm setting for using an endmill in a milling machine.
5. Calculate the proper feedrate for milling operations.

Put a check beside the statement that is true for you.

☐ **I can do these tasks and want to go through the Performance Checklist.**

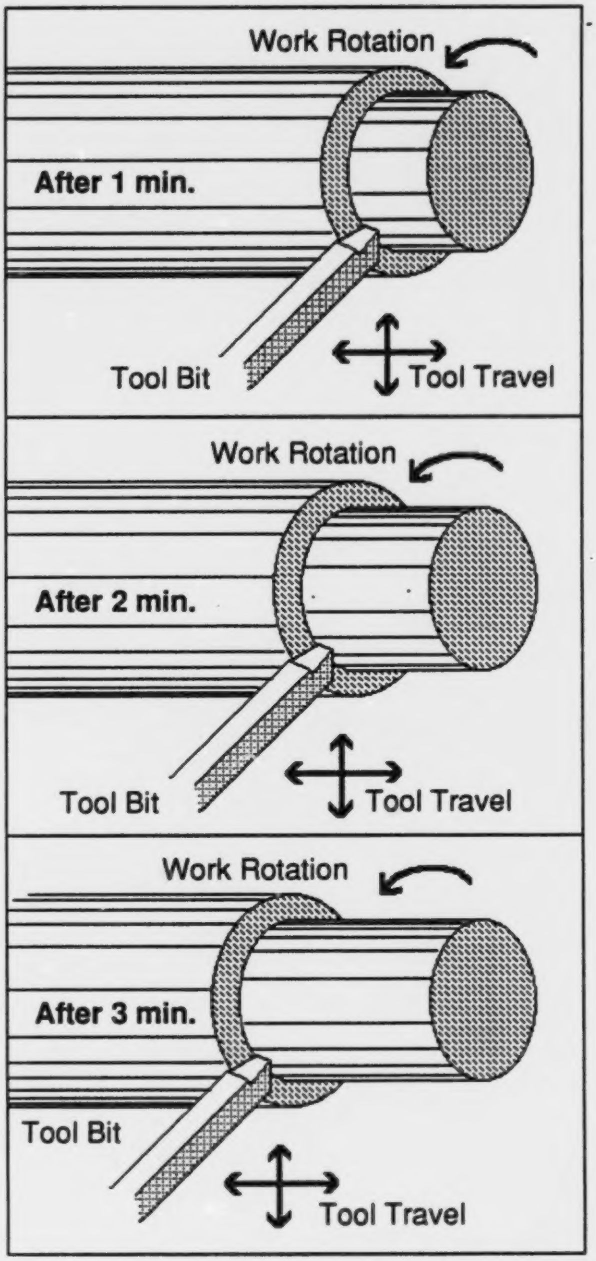
SEE YOUR INSTRUCTOR

☐ **I want to study the information in this unit before I do the Performance Checklist.**

TURN PAGE AND BEGIN

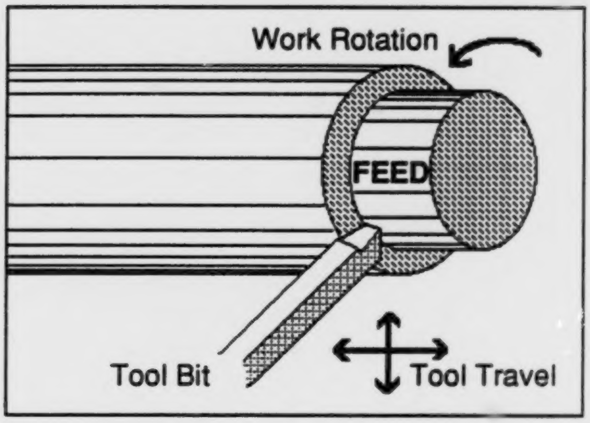
A. TERMS

1. **Cutting speed** - the linear (straight line) distance a point on the circumference (outside) of a rotating tool or workpiece will travel in one minute. Cutting speed is expressed as either meters per minute (mpm) or feet per minute (fpm). It may also be referred to as surface speed and be expressed as surface feet per minute (sfpm). Cutting speed is directly related to revolutions per minute (rpm) and the diameter of the rotating workpiece or tool.

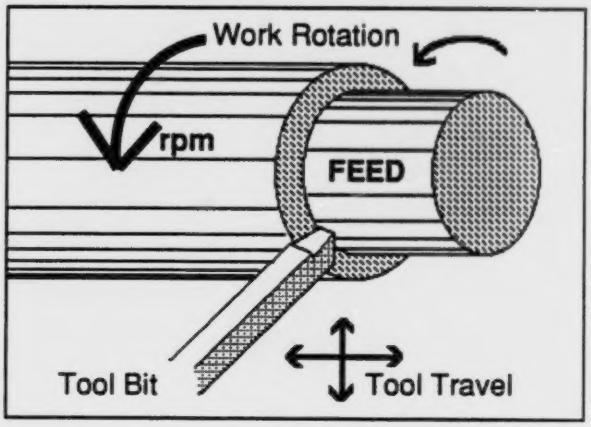

Work Rotation
After 1 min.
Tool Bit
Tool Travel
Work Rotation
After 2 min.
Tool Bit
Tool Travel
Work Rotation
After 3 min.
Tool Bit
Tool Travel

2. **Feed** (feedrate) - movement

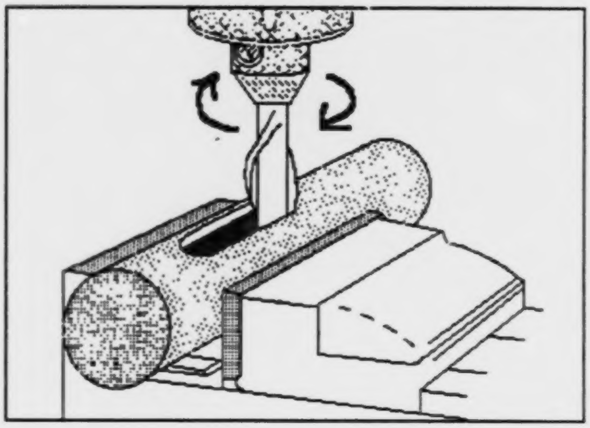
of the tool past the workpiece. Feed is expressed as meters per minute (mpm) or inches per minute (ipm). Feed determines how much material is removed from the workpiece in a given amount of time.


Work Rotation
FEED
Tool Bit
Tool Travel

3. **Revolutions per minute (rpm)** - the number of times a workpiece or tool rotates (makes a complete turn) in one minute.

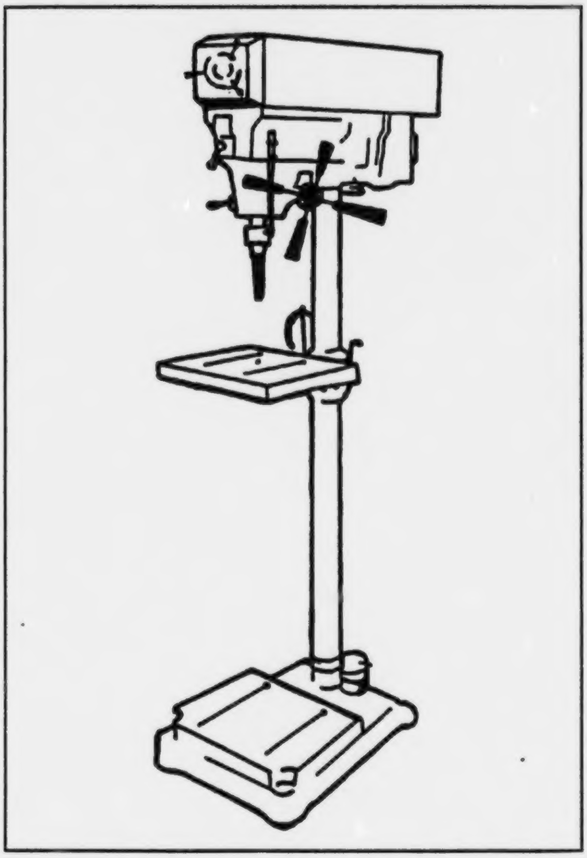

Work Rotation
rpm
FEED
Tool Bit
Tool Travel

4. **Spindle speed** - the rotating speed of the spindle on a machine tool. Spindle speed is expressed in revolutions per minute (rpm).



5. **Lands**--largest diameter of the cutting portion of a drill.

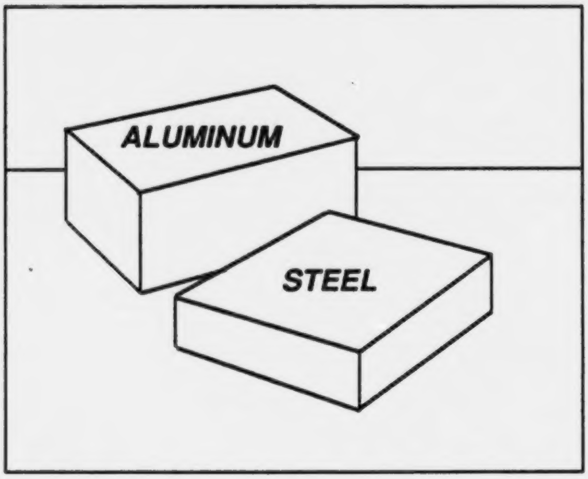
B. FACTORS THAT INFLUENCE MACHINING FEEDS AND SPEEDS



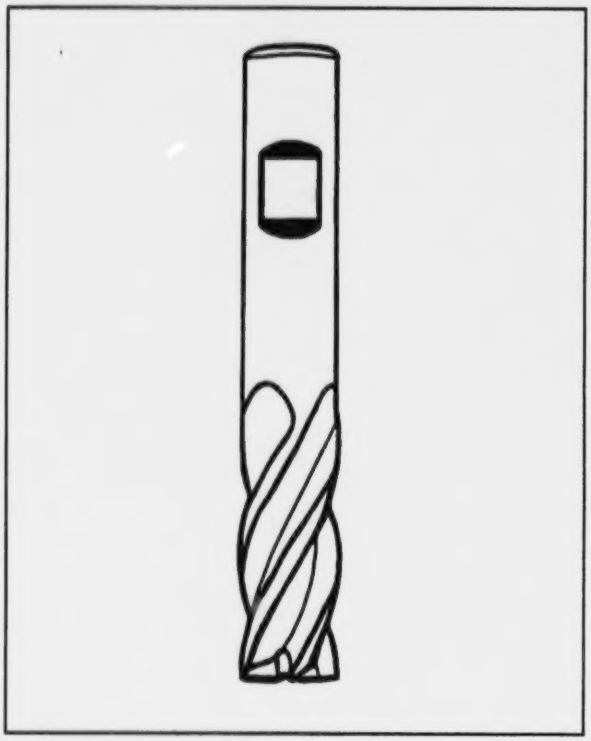
This unit will teach you how to figure the best speed and feed settings for drilling, turning (lathework) and milling. The numbers you get are based on laboratory conditions. You must adjust them for each particular machining job.

Factors affecting feed and speed:

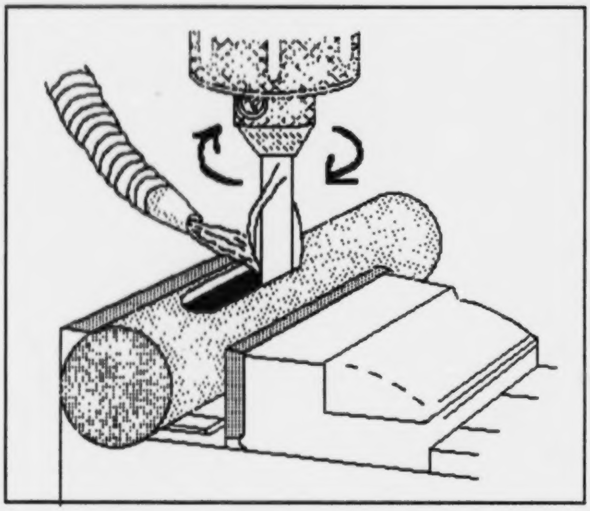
1. **Workpiece material.** You can machine softer materials at higher speeds than hard materials. For example, you can usually run aluminum at higher speeds than you can run steel.

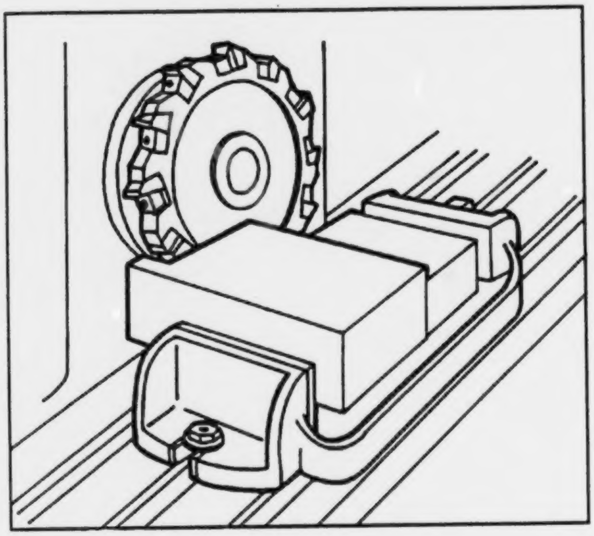


2. **Cutting tool material.** You can operate some tool materials at higher speeds and feeds (and higher temperatures) than others. For instance, you can run high speed (alloy) steel at higher speeds than you would use for carbon steel. You can run carbide faster than high-speed steel. You can run some ceramic tools faster than carbide.

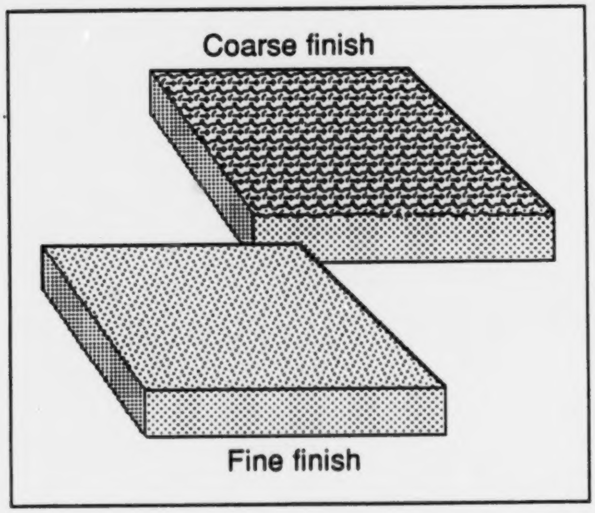


3. **Cutting fluids.** You mist or flood cutting fluids onto the tool to remove heat from the cutting area. Use of cutting fluids will allow you to run the tool at much higher speeds.





4. **Rigidity of the setup.** If the machine is well maintained and "tight," and the tool and workpiece are held firmly, you can run the tool at higher feeds. This allows you to remove more material in less time.



5. **Finish.** When you want a fine finish, you should remove less material, but you may actually increase speeds.

DO SKILL SHEET 1

CALCULATE FEEDS AND SPEEDS

SKILL SHEET 1

Part 1. Write the term that goes with each definition below.

Revolutions per minute (rpm)
Feed
Cutting speed
Spindle speed

1. The rotating speed of the spindle on a machine tool is the________

2. The movement of the tool past the workpiece is the____________

3. The number of times a workpiece or tool rotates in one minute is ______________

4. The distance a point on the outside edge of a rotating tool will travel in one minute is_____________

Part 2. For each situation described below write "increased" or "decreased" to describe whether feeds and speeds should go up or down.

______ 5. You will machine a soft material such as aluminum.

______ 6. You will switch cutting tool material from carbide to high-speed steel.

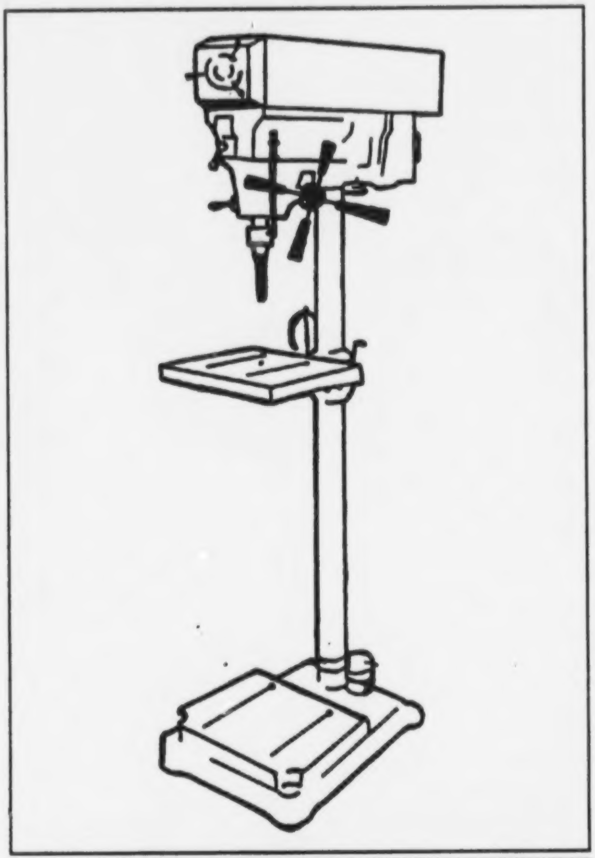
______ 7. You will flood the tool with a proper cutting fluid during cutting.

______ 8. The workpiece is clamped very lightly to prevent damage.

______ 9. You are using an old machine with worn parts.

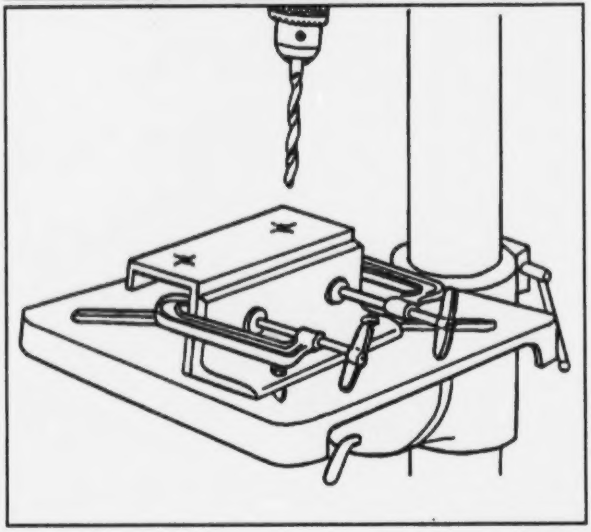
______ 10. You need a fine finish.

C. CALCULATING SPINDLE SPEED FOR DRILLING



When you use a drilling machine, the workpiece is held firmly to the drilling table or in a vise. The rotating (turning) cutting tool (usually a twist drill) is pushed into the surface of the workpiece to create a hole.

Because the drill is held in the spindle of the drillpress, the important calculation for drilling is the spindle speed. You must figure the proper revolutions per minute (rpm) setting for the spindle.

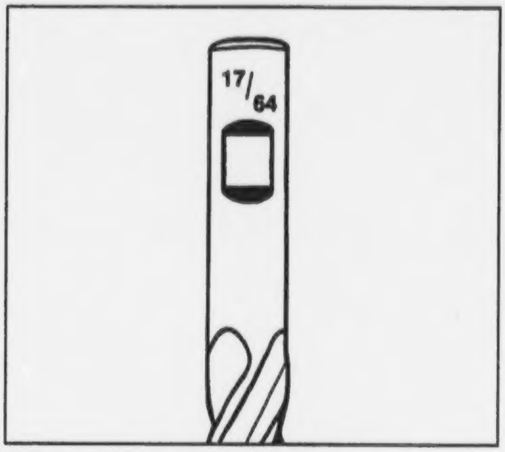


To figure the **revolutions per minute** (rpm), you need two pieces of information:

1. the diameter of the drill
2. the appropriate cutting speed for the material you will be drilling.

You find the diameter of the drill by looking at the markings on the base of the drill. You can also measure across the "**lands**" (largest diameter of the cutting portion) with a micrometer or vernier caliper.

You find cutting speed values on standard tables. These speeds have been determined through years of experimenting. You can rely on them.



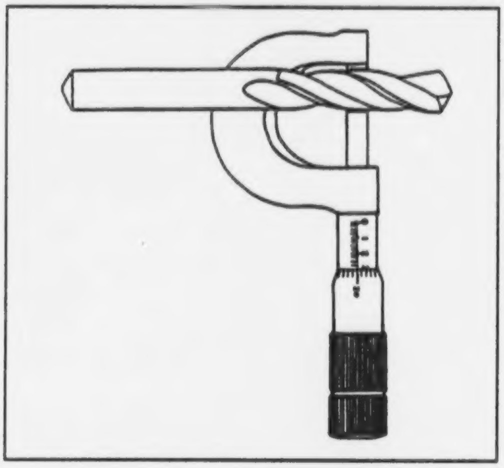


Table of Cutting Speeds

Material	Meters Per Minute	Feet Per Minute
Low-carbon steel (0.05-0.30% carbon)	24.4-33.5	80-110
Medium-carbon steel (0.30-0.60% carbon)	21.4-24.4	70-80
High-carbon steel (o.60-1.70% carbon)	15.2-18.3	50-60
Steel forgings	15.2-18.3	50-60
Alloy steel	15.2-21.4	50-70
Stainless steel	9.1-12.2	30-40
Cast iron, soft gray	30.5-45.7	100-150
Cast iron, hard-chilled	21.4-30.5	70-100
Cast iron, malleable	24.4-27.4	80-90
Ordinary brass and bronze	61.0-91.4	200-300
High-tensile bronze	21.4-45.7	70-150
Monel metal	12.2-15.2	40-50
Aluminum and its alloys	61.0-91.4	200-300
Magnesium and its alloys	76.2-122.0	250-400
Slate, marble, and stone	4.6-7.6	15-25
Bakelite and similar plastics	30.5-45.7	100-150
Wood	91.4-122.0	300-400

Here is the inch formula you will need:

$$\frac{\text{Cutting speed X 12}}{\text{Drill Diameter X pi}} = \text{spindle rpm}$$

A simplified formula for cutting speed in the inch measuring system cancels out pi. (It lets pi = 3 instead of 3.1416).

$$\frac{\text{Cutting speed X 4}}{\text{Drill diameter X pi}} = \text{spindle rpm}$$

For example:

You need to drill a 1/2" hole in a piece of mild steel. From the table you determine that the cutting speed for low-carbon steel is 110 feet per minute. Using the formula:

$$\frac{110 \text{ X } 4}{.5 \text{ X } 1} = \frac{440}{.5} = 880 \text{ rpm}$$

The metric formula is a little different:

$$\frac{\text{Cutting speed X 1000}}{\text{Diameter X pi}} = \text{rpm setting drill}$$

For an example using metric units, consider drilling an 18 mm hole in stainless steel. The cutting speed for stainless steel from the table is 40 mpm. Using the formula:

$$\frac{40 \text{ X } 1000}{18 \text{ X } 3.1416} = \frac{4000}{56.5488} = 707 \text{ rpm}$$

DO SKILL SHEET 2

DO ACTIVITY SHEET 1

CALCULATE SPEEDS AND FEEDS

SKILL SHEET 2

1. Write the formulas for calculating spindle rpm for drilling:

   A. Inch system

   B. Metric system

2. Solve the following using the appropriate formula and the cutting speeds table. Show the steps you take to figure the math.

   A. Drill a 3/4-inch hole in aluminum. (Cutting speed = 250sfpm)

Another problem is on the back of this page.

B. Drill a 8 mm hole in magnesium. (Cutting speed = 100mpm)

CALCULATE SPEEDS AND FEEDS

ACTIVITY SHEET 1

Demonstrate to your instructor that you can set your drill press to the following rpm settings. (Note: if belt changing is required, set to closest speed available).

1. 60 rpm
2. 500 rpm
3. 325 rpm
4. 800 rpm

Using the metric formula:

$$\frac{\text{Cutting speed X 1000}}{\text{Workpiece diameter X pi}} = \text{spindle rpm}$$

Take a finish cut on a 20 mm diameter workpiece made of brass using a high-speed steel tool. The cutting speed for brass is 53 mpm.

$$\frac{53 \text{ X } 1000}{20 \text{ X } 3.1416} = \frac{53{,}000}{62.832} = 844 \text{ rpm}$$

DO SKILL SHEET 3

DO ACTIVITY SHEET 2

CALCULATING SPEEDS AND FEEDS

SKILL SHEET 3

1. Write the formulas for calculating lathe spindle rpm:

   A. Inch system

   B. Metric system

2. Solve the following problems (find the rpm). Show your work.

   Use the table on page 19.

   A. Finish turn a 5-inch-diameter aluminum workpiece.

Another problem is on the back.

B. Rough turn a 13-mm-diameter cast iron workpiece

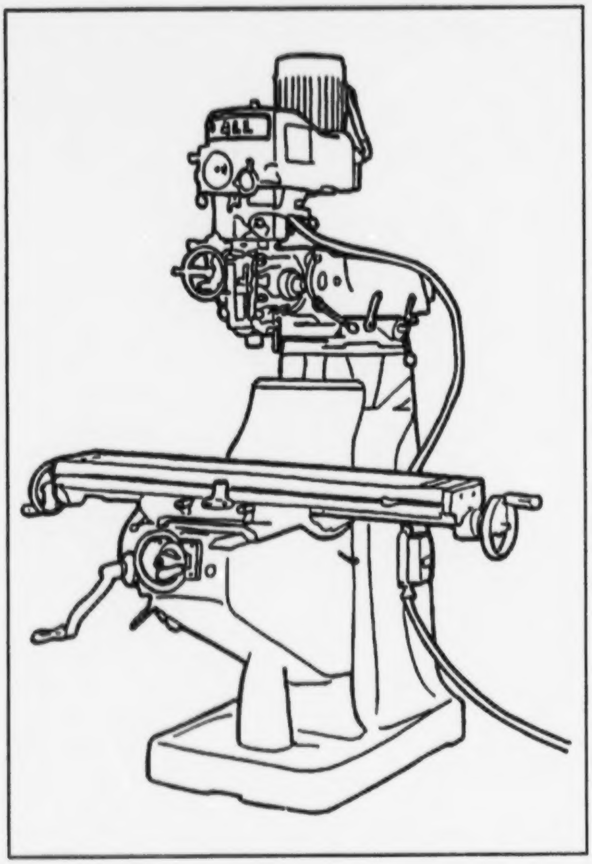
CALCULATING SPEEDS AND FEEDS

ACTIVITY SHEET 2

Demonstrate to your instructor that you can set your lathe spindle to run at the following rpm settings. (Note: if a variable speed drive is not present, set as closely as your lathe gearing or belts will permit.)

1. 75 rpm
2. 250 rpm
3. 375 rpm
4. 800 rpm

IV. CALCULATING SPINDLE SPEED FOR MILLING



Calculation of spindle speed rpm settings for milling is identical to drilling. Spindle rpm is set to control the speed of the rotating milling cutter. However, cutting speeds from the milling table below are somewhat lower than those used for drilling. The formulas are identical:

Table of Cutting Speeds

Material	Cutting Speed Range	
	mpm*	fpm**
Low-carbon steel	18.3-24.4	60-80
Medium-carbon steel	18.3-24.4	60-80
High-carbon steel	15.2-21.4	50-70
Tool steel	15.2-21.4	50-70
Stainless steel	15.2-21.4	50-80
Grey cast iron	15.2-21.4	50-80
Malleable iron	24.4-30.5	80-100
Aluminum and its alloys	121.9-304.8	400-1000
Brass	61.0-91.4	200-300
Bronze	30.5-61.0	100-200

*meters per minute
**feet per minute

Inch formula

$$\frac{\text{Cutting speed X 12}}{\text{Cutter diameter X pi}} = \text{spindle rpm}$$

Or simplified:

$$\frac{\text{Cutting speed X 4}}{\text{Cutter diameter}} = \text{spindle rpm}$$

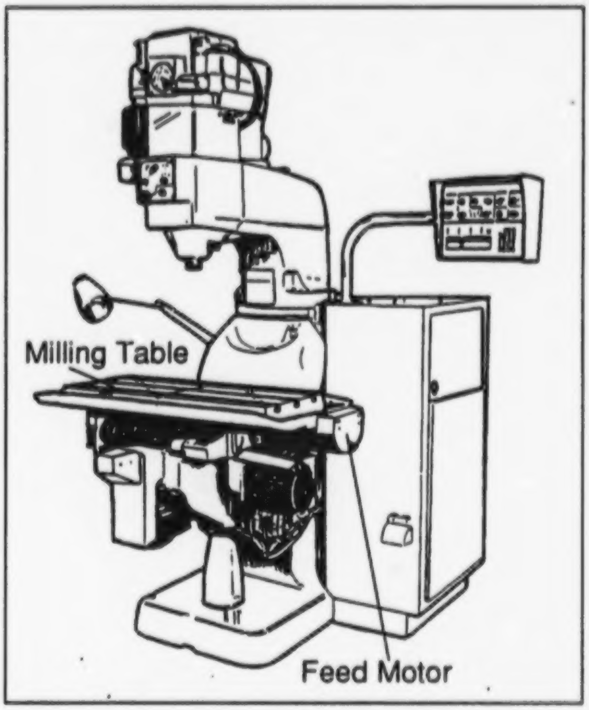
Metric Formula

$$\frac{\text{Cutting speed X 1000}}{\text{Setting cutter diameter X pi}} = \text{spindle rpm}$$

Calculate the spindle rpm setting for 15mm endmill high carbon steel. Cutting speed is 16 mpm.

$$\frac{16 \text{ X } 1{,}000}{15 \text{ X } 3.1416} = \frac{16{,}000}{47.124} = 340 \text{ rpm}$$

When a milling machine is equipped with a power feed device or computer control, you must also calculate the proper feedrate. This is the rate of movement of the milling table past the rotating tool.



Feedrate is expressed in inches per minute or millimeters per minute.

You need three pieces of information to calculate feedrate.

1. Calculate the spindle rpm as previously shown.
2. Use another table find the "feed per tooth" amount from the

"Feed-per Tooth" table.

This table corresponds to the cutter type and the material to be machined.

Material	Plain Mills		Face Mills	End Mills	Form Relieved Mills	Slitting Saws
	Heavy Duty	Light Duty & Side Mills				
Low-carbon steel, free machining	0.25 (0.010)*	0.15 (0.006)	0.31 (0.012)	0.15 (0.006)	0.10 (0.004)	0.08 (0.002)
Low-carbon steel	0.20 (0.008)	0.13 (0.005)	0.25 (0.010)	0.13 (0.005)	0.08 (0.003)	0.08 (0.003)
Medium-carbon steel	0.20 (0.008)	0.13 (0.005)	0.23 (0.009)	0.10 (0.004)	0.08 (0.003)	0.05 (0.002)
High-carbon steel	0.10 (0.004)	0.08 (0.003)	0.15 (0.006)	0.005 (0.002)	0.05 (0.002)	0.05 (0.002)
Stainless steel, free machining	0.20 (0.008)	0.13 (0.005)	0.25 (0.010)	0.10 (0.004)	0.08 (0.003)	0.05 (0.002)
Stainless steel	0.10 (0.004)	0.08 (0.003)	0.15 (0.006)	0.05 (0.002)	0.05 (0.002)	0.05 (0.002)
Cast iron, soft	0.31 (0.012)	0.20 (0.008)	0.36 (0.014)	0.20 (0.008)	0.10 (0.004)	0.10 (0.004)
Cast iron, medium	0.25 (0.010)	0.15 (0.006)	0.31 (0.012)	0.15 (0.006)	0.10 (0.004)	0.08 (0.002)
Malleable iron	0.25 (0.010)	0.15 (0.006)	0.31 (0.012)	0.15 (0.006)	0.10 (0.004)	0.08 (0.002)
Brass and bronze, medium	0.25 (0.010)	0.20 (0.008)	0.33 (0.013)	0.15 (0.006)	0.10 (0.004)	0.08 (0.003)
Aluminum and its alloys	0.41 (0.016)	0.25 (0.010)	0.51 (0.020)	0.25 (0.010)	0.18 (0.007)	0.10 (0.004)

*millimeters (inches)

3. Count the number of teeth on the cutter.

You now have all the information necessary to calculate feedrate.

The formula looks like this:

F = R x T x RPM

F = feedrate

R = feed per tooth

T = number of teeth on the cutter

RPM = spindle rpm

Calculate the feedrate for the following conditions:

You are to machine a block of low carbon steel with a 1" diameter four-flute (tooth) endmill made of high-speed steel. Cutting speed is 60 fpm.

First calculate spindle rpm (RPM)

$$\frac{60 \text{ X } 12}{1 \text{ X } 3.1416} = \frac{720}{3.1416} = 229 \text{ rpm}$$

Find the feed per tooth (R) from

the table: = .006 inch

Count the number of teeth: = 4

F = R x T x RPM

F = .006 x 4 x 229

F = 5.496 inches per minute
(ipm) of feed

DO SKILL SHEET 4

CALCULATE SPEEDS AND FEEDS

SKILL SHEET 4

1. When is it necessary to calculate the feedrate for milling?

2. How is feedrate expressed?

3. What three pieces of information do you need to calculate feedrate?

4. Calculate the feedrate (inches per minute) for the following case:

   Machine an aluminum workpiece (rough cutting) with a 1/2" two-flute (tooth) endmill. Your answer should be in inches/minute.

CALCULATE SPEEDS AND FEEDS

UNIT TEST

Part 1. Match the terms below with their definitions.

A. Cutting speed
B. Feed
C. Revolutions per minute
D. Spindle speed

___ 1. Movement of the cutting tool past the workpiece.
___ 2. The distance a point on the circumference of a rotating tool will travel in one minute.
___ 3. The rotating speed of the spindle on a machine tool.
___ 4. The number of times a workpiece or tool rotates in one minute.

Part 2. Arrange the cutting tool materials listed below in order from slowest to fastest. Put a 1 for the slowest and a 4 for the fastest, etc.

___ 5. Carbon steel
___ 6. Carbide
___ 7. Ceramic
___ 8. High-speed steel

Part 3. Answer the following questions:

9. What can you do to remove heat from the cutting tool and workpiece during machining?

10. What two pieces of information are needed to calculate the proper spindle RPM setting for drilling?

A.

B.

11. Calculate the proper spindle RPM for drilling a 3/4" hole in a block of aluminum. (Cutting speed = 200)

12. Calculate the proper spindle rpm setting for finish turning a 3' diameter workpiece made of stainless steel. (Cutting speed = 100)

13. Calculate the proper spindle rpm setting for milling grey cast iron at 15 mph with a 30mm endmill.

14. What three pieces of information do you need to calculate the proper feedrate for milling?

    A.

    B.

    C.

15. Calculate the feedrate for milling brass with a 1/4 two-flute endmill.

CALCULATE FEEDS AND SPEEDS

PERFORMANCE CHECKLIST

Student____________________________has successfully performed the following tasks:

	YES	NO	COMMENTS
1. Identified factors that determine speeds and feeds.			
2. Calculated the proper rpm setting for drilling on a drill press.			
3. Calculated the proper rpm setting for turning a workpiece in a lathe.			
4. Calculated the proper rpm setting for using an endmill in a milling machine.			
5. Calculated the proper feedrate for milling operations.			

Satisfactory - Should move on ☐

Repeat this unit ☐

____
Student's Signature/Date

____
Instructor's Signature/Date

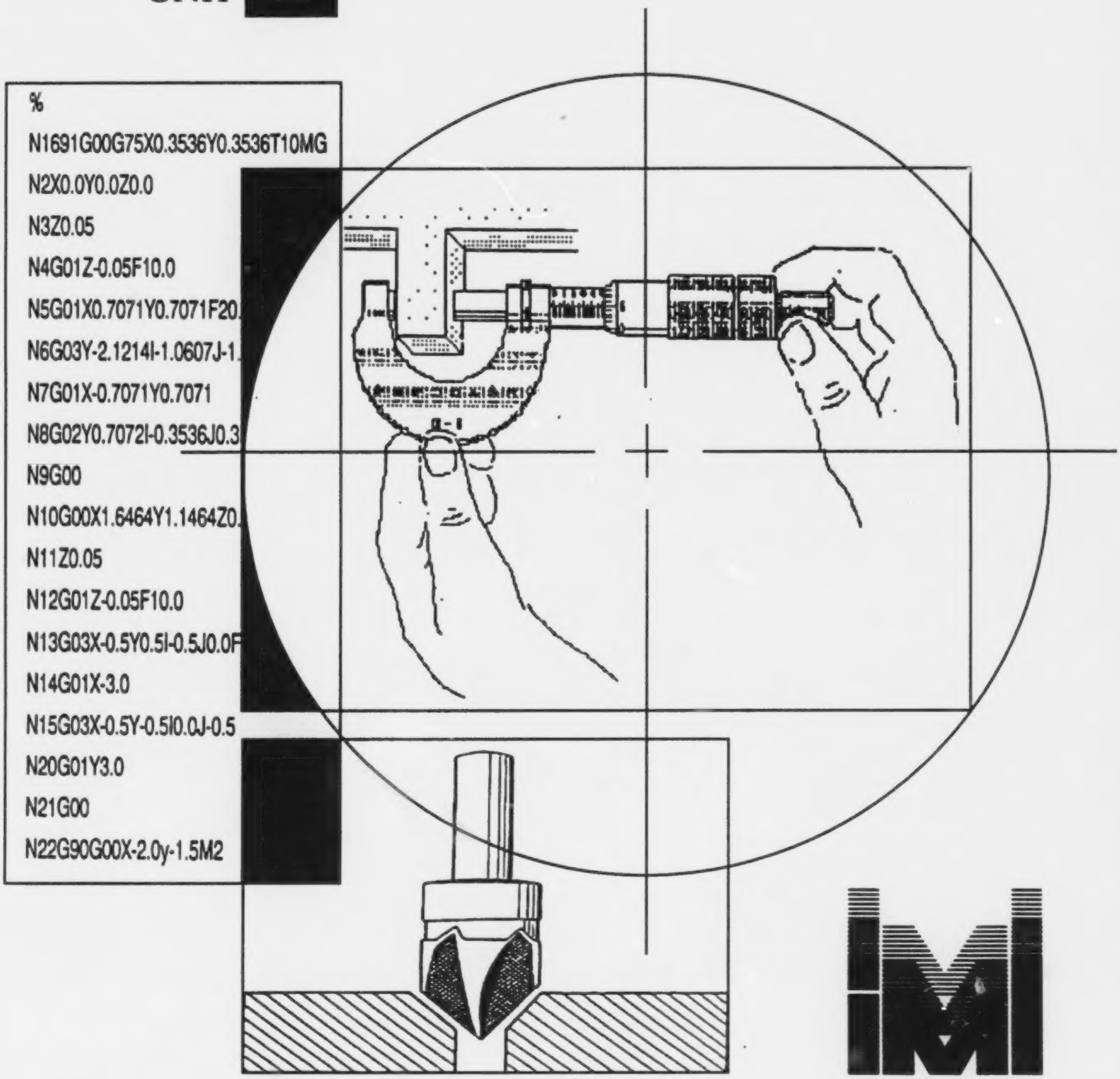
60

Machine Shop

90-9200-S
SUPPLEMENTARY UNITS

USING A RULE

UNIT 2



```
%
N1691G00G75X0.3536Y0.3536T10MG
N2X0.0Y0.0Z0.0
N3Z0.05
N4G01Z-0.05F10.0
N5G01X0.7071Y0.7071F20.
N6G03Y-2.1214I-1.0607J-1.
N7G01X-0.7071Y0.7071
N8G02Y0.7072I-0.3536J0.3
N9G00
N10G00X1.6464Y1.1464Z0.
N11Z0.05
N12G01Z-0.05F10.0
N13G03X-0.5Y0.5I-0.5J0.0F
N14G01X-3.0
N15G03X-0.5Y-0.5I0.0J-0.5
N20G01Y3.0
N21G00
N22G90G00X-2.0y-1.5M2
```

USING A RULE

INTRODUCTION

In all operations as a machinist, you will use a pocket rule for measuring. The pocket rule is as important as food to the machinist's well being. Accuracy is important. A machine piece must be the right size. In this unit you will learn to use both rigid and flexible rules.

UNIT OBJECTIVE

After completing this unit, you should be able to use the pocket rule to measure correctly. You also should be able to care for the steel rule.

SPECIFIC OBJECTIVES

After completing this unit, you should be able to:

1. Describe the care of the steel rule.
2. Define the terms used for the steel rule.
3. Identify the various ways to measure with the various rules.
4. Make measurements to the 1/64th of an inch.

Put a check beside the statement that is true for you.

☐ I can do these tasks and want to go through the Performance Checklist.

SEE YOUR INSTRUCTOR

☐ I want to study the information in this unit before I do the Performance Checklist.

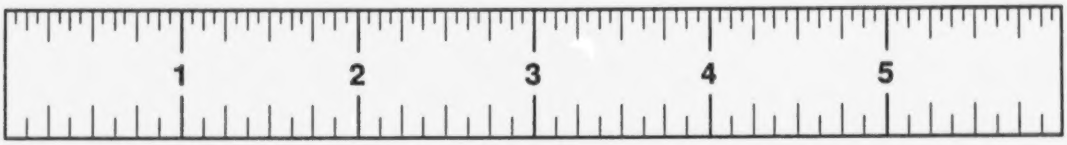
TURN PAGE AND BEGIN

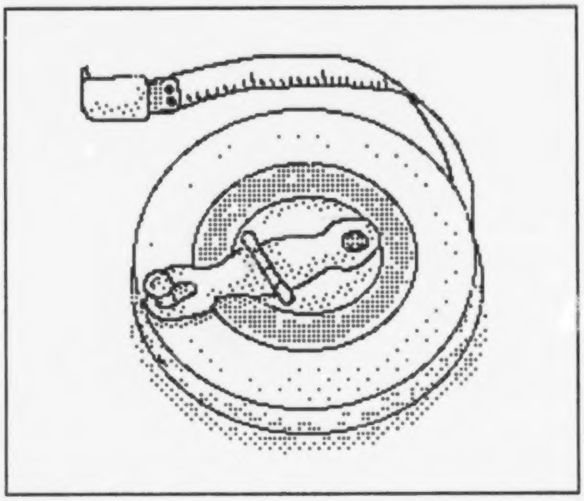
A. TERMS

1. **Linear measurement** - measuring along a straight line.
2. **Rigid** - stiff, will not bend.
3. **Flexible** - will bend.
4. **Graduations** - marks on the rule (flexible and/or rigid).
5. **Align** - to bring into line.

B. EQUIPMENT

1. Rigid rule - a strip of brass or lead with at least one edge marked with units of measurement.





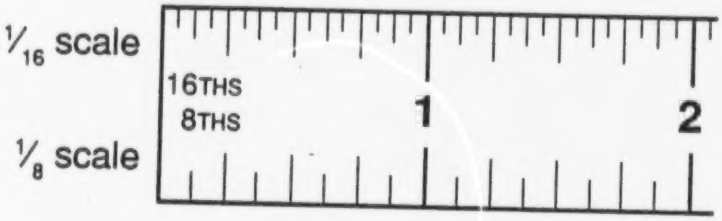
2. Flexible rule (tape) - same as a rigid rule but can be rolled up.

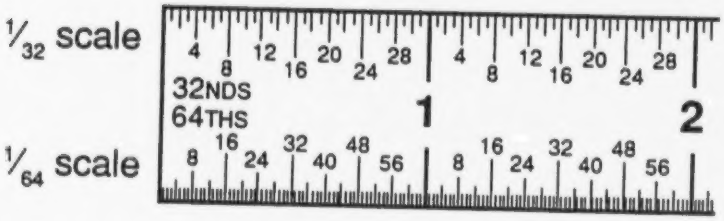
C. CARE OF THE STEEL RULE

1. Keep the rule clean, free of oil, chips, or layout dye.
2. Do not use the rule for cleaning chips out of a machine or part.
3. Keep rule away from magnets. A magnetized rule will hold steel chips and cause measuring errors.
4. Be careful how you use the rule, where and how you lay it down. It can become worn, marred, or scratched.
5. Keep away from revolving cutters or parts.
6. Never bend a rigid rule or over bend a flexible rule.
7. Do not stamp your initials or name on the rule. It will become distorted.

D. READING THE STEEL RULE

The terms scale and graduation mean the same thing in shop work. They mean the marks that give you the inch or a part of an inch.


1/16 scale
16THS
8THS
1/8 scale
1
2


1/32 scale
4
8
12
16
20
24
28
32NDS
64THS
1
2
1/64 scale
8
16
24
32
40
48
56

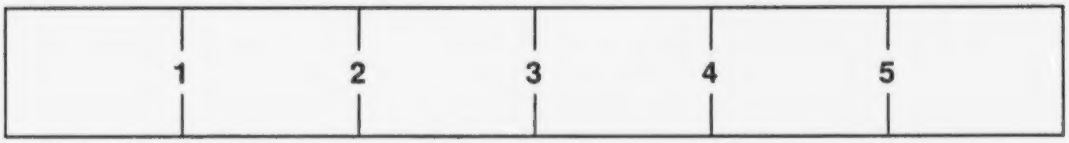
The rule usually has four sets of the scales, one on each edge of the front and back. These scales are 1/64", 1/32", 1/16" and 1/8".

Now get your rule and look at it.

The scale looks the same but might not be as easy to read as the picture on this page. Try to read your rule. You will be working with it in the machine shop.

1. 1-INCH SCALE

Let's look how the rule is broken into the scales and how the scales are divided. The rule is divided into inches first (1 inch, 2 inches, 3 inches, 4 inches, etc.) These large numbers are in the center of the rule.



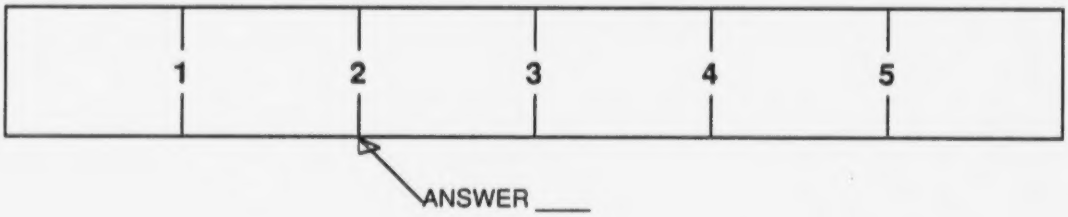
DO SKILL SHEET 1

USING A RULE

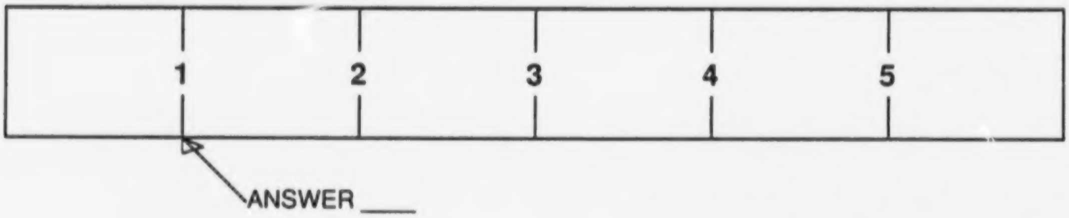
SKILL SHEET 1

To make sure you understand about the inches let's read a few measurements. The arrow (↑) points to the figure you are to read. Write the answer in the space provided.

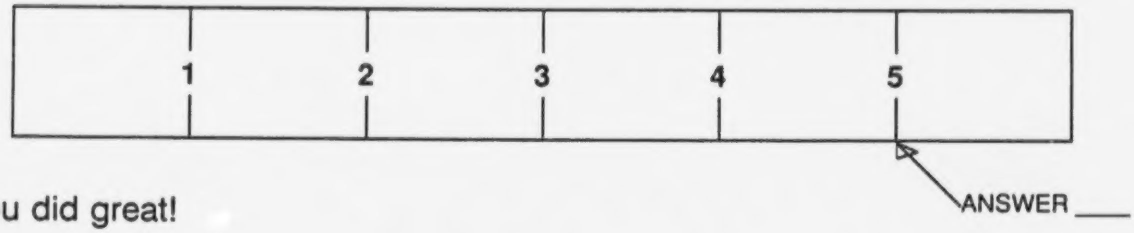
1.


1
2
3
4
5
ANSWER ____

2.


1
2
3
4
5
ANSWER ____

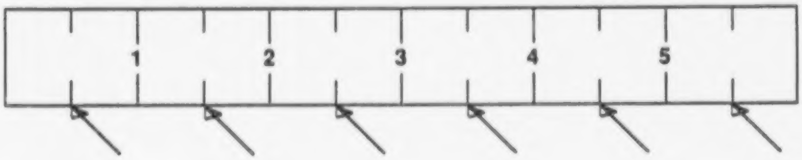
3.


1
2
3
4
5
ANSWER ____

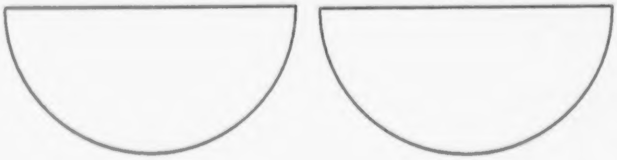
You did great!

2. 1/2-INCH SCALE

Now let's try to get a little closer measurement. You notice in the picture that the rule not only has the inch marked but also has another graduation. That graduation is the one-half inch mark. Look at your rule. Can you find the half inch?



The one-half is a fraction written 1/2. If we put the two one-halves on top of each other, they will be equal.



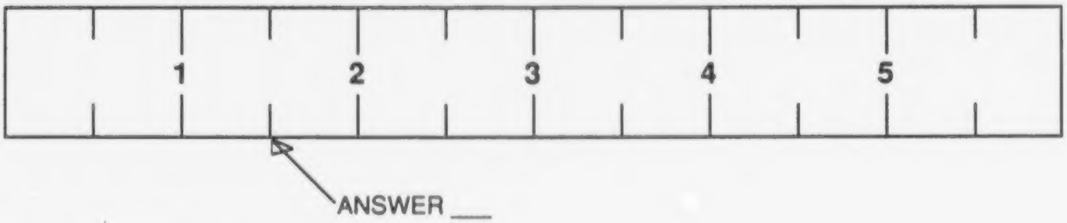
DO SKILL SHEET 2

USING A RULE

SKILL SHEET 2

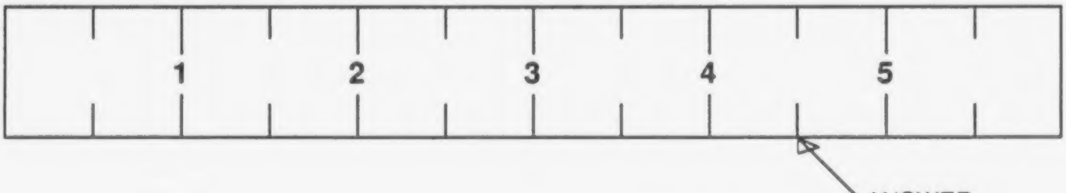
Let's read some measurements. Write your answer on the space provided. Check with your instructor. The arrows point to the one-half graduations. Remember, if you have passed an inch graduation, your answer must include the inch plus the one half.

1.



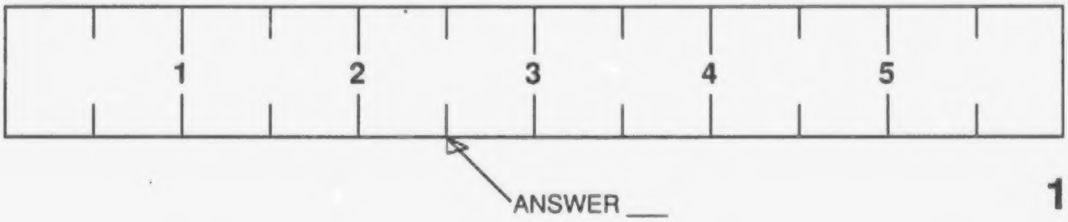
ANSWER ___

2.



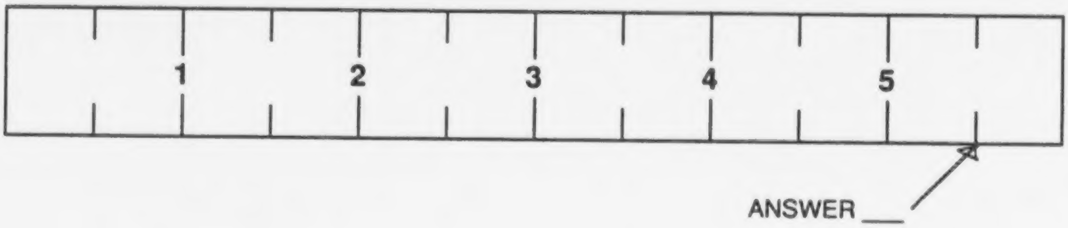
ANSWER ___

3.



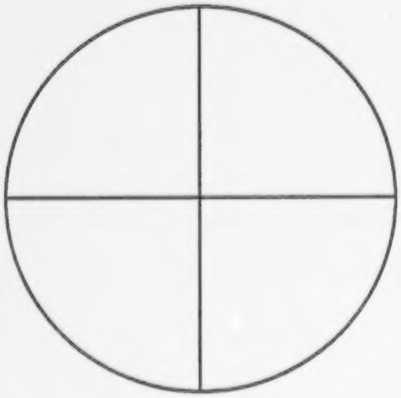
ANSWER ___

4.


1
2
3
4
5
ANSWER ___

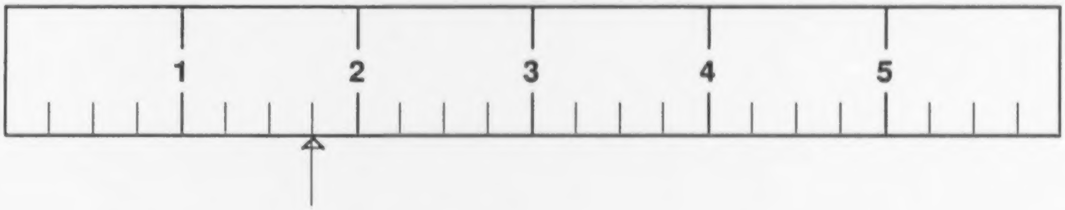
I'm so proud of you! You are doing really well.

3. 1/4-INCH SCALE



Now let's divide the one-half into two parts (or the one inch into four parts--another fraction). This fraction is one-fourth (1/4th). When we put all four of these sections together, they equal one inch.

Look at the rule below. The arrow is pointing to a mark past the large number 2 on the rule. We know that the measurement will be two inches plus something. Count the marks past the 2 to the arrow. Good! There are three. We have 2 inches plus 3/4ths of another inch. The reading is 2 and 3/4ths inches.



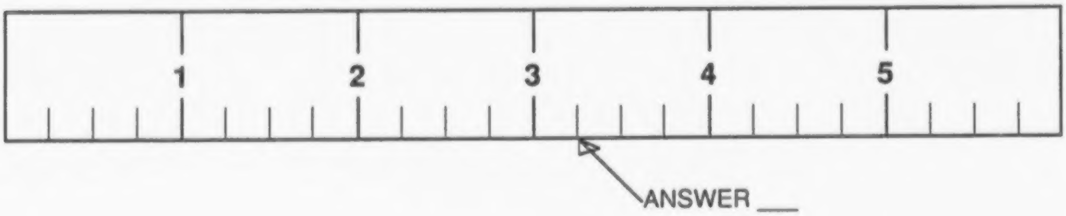
DO SKILL SHEET 3

USING A RULE

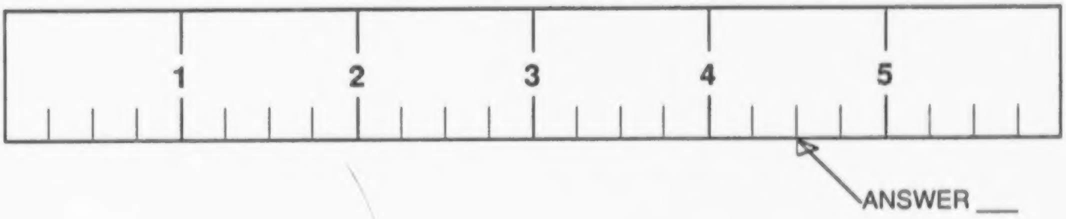
SKILL SHEET 3

Read the measurements below and write your answers in the spaces provided. Check with your instructor.

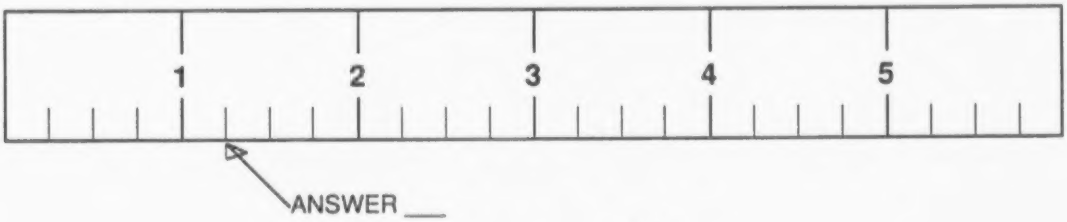
1.


1
2
3
4
5
ANSWER ___

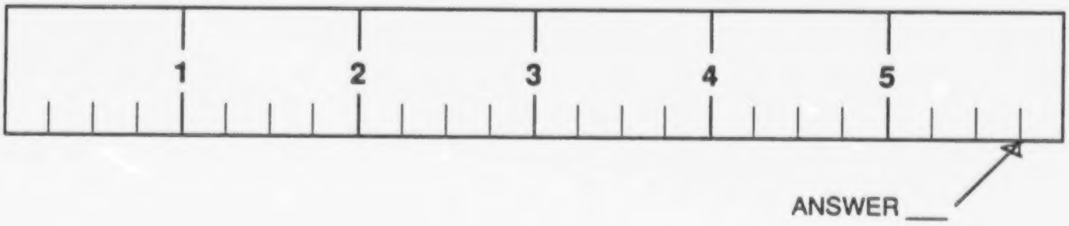
2.


1
2
3
4
5
ANSWER ___

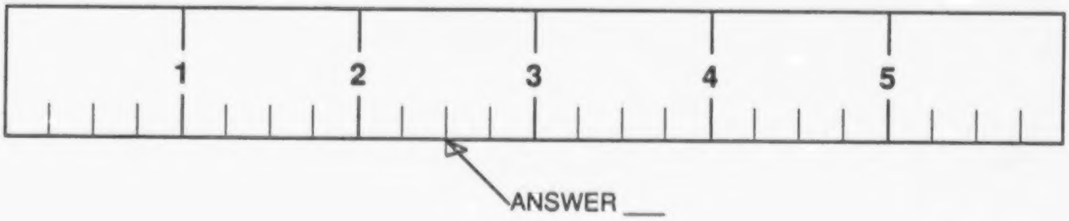
3.


1
2
3
4
5
ANSWER ___

4.


1
2
3
4
5
ANSWER ___

5.

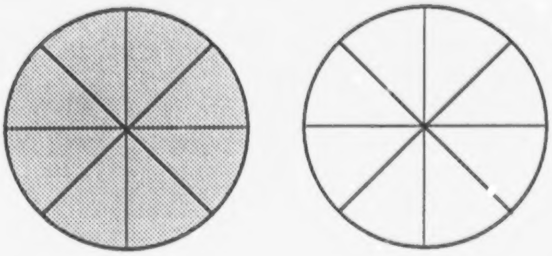

1
2
3
4
5
ANSWER ___

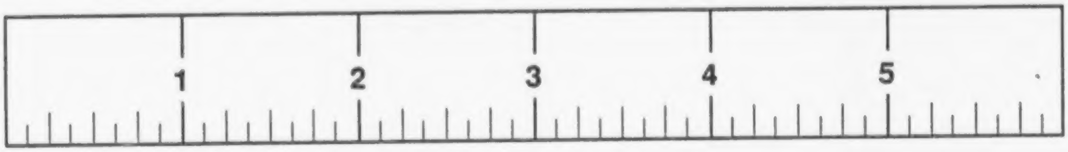
You are doing super!

3. 1/8-INCH SCALE

What comes next? We will now need to divide the inch into eighths.

Let's look at the box. There are eight sections to this box. Well, our rule also has equal scales on it. This scale marks parts of an inch. The marks stand for one-eighth of an inch (1/8).





Look how the marks change in size as the fraction changes. The 1/4-inch marks are longer than the 1/8-inch marks. If you keep this in mind, it will help you in figuring the fraction.

DO SKILL SHEET 4

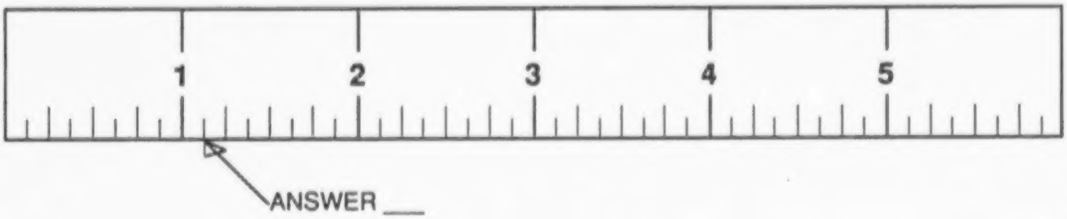
USING A RULE

SKILL SHEET 4

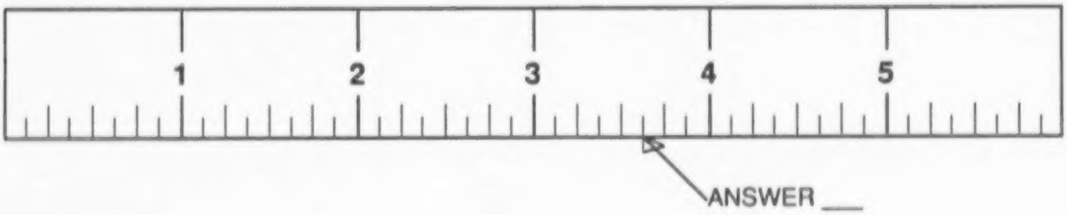
Read the measurements on the following rule where the arrow points.

Write your answers on the spaces provided. Check with your instructor.

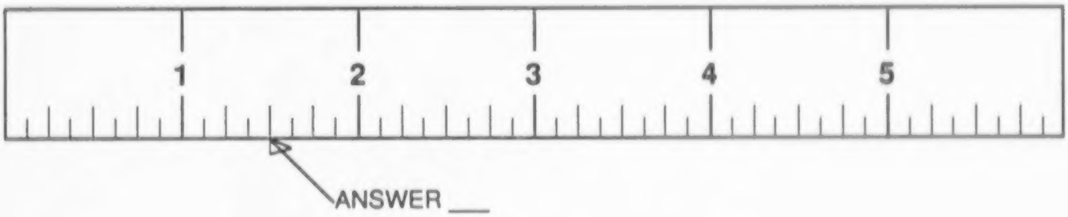
1.


1
2
3
4
5
ANSWER ___

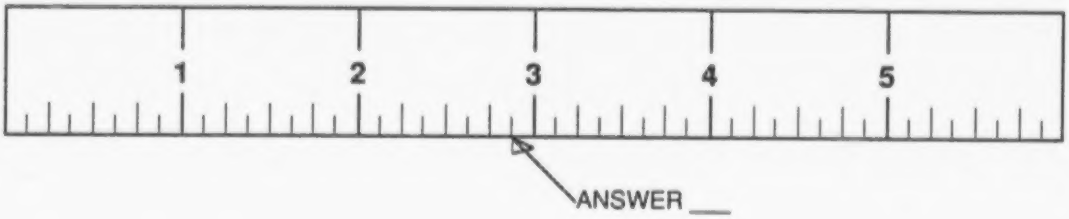
2.


1
2
3
4
5
ANSWER ___

3.

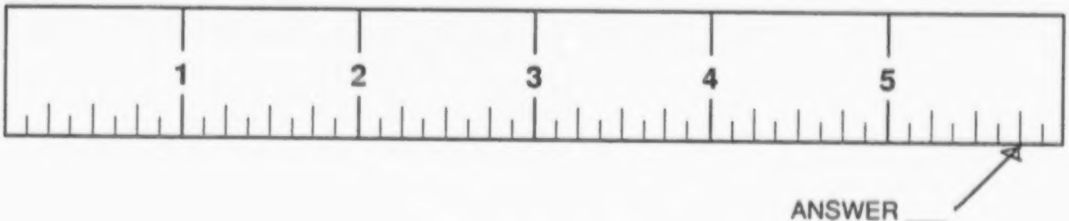

1
2
3
4
5
ANSWER ___

4.



ANSWER ___

5.



ANSWER ___

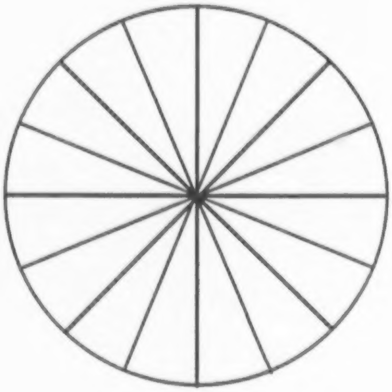
Your progress is fantastic!

4. 1/16-INCH SCALE

Now we're really making progress. After dividing the quarters into eighths, we will divide the eighths into 16ths. Then we'll divide the 16ths into 32nds, and last, the 32nds into 64ths.

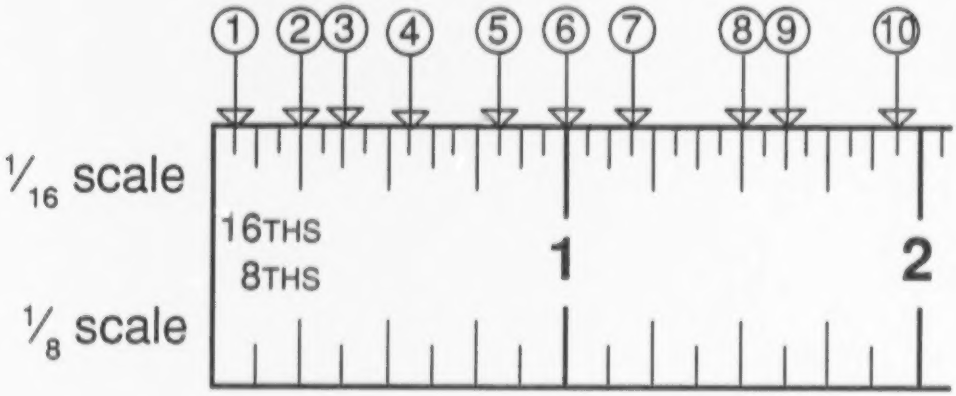
Because the fractions are smaller and smaller now, we will show the rule larger than its actual size. This will let you see the divisions more easily.

Let's look at the rule divided into 16ths. This means the eighth has been divided into half to make 1/16th of an inch. The inch now has 16 parts. Look at the pie. It is cut into 16 equal pieces.



Remember that the marks for the 1/16th are shorter than the marks that stand for 1/8th.

Below is a rule marked using 1/16th. (Remember, we are using a larger size than the rule you will have with your tools). Look at your rule. Can you find the 1/16th marks.



①
②
③
④
⑤
⑥
⑦
⑧
⑨
⑩
1/16 scale
16THS
8THS
1
2
1/8 scale

Count up to the arrow (↑) number. If it passes a whole (large number marked on the rule) number, that

79

whole number will be in the answer. Whatever the number of lines is after the whole number will be put over 16 to make the fraction a 16th.

Sometimes you will need to reduce the number to the smallest fraction. For example, if your answer is 6/16ths, will read as 3/8; 4/16 is also 1/4, and 10/16 will read 5/8.

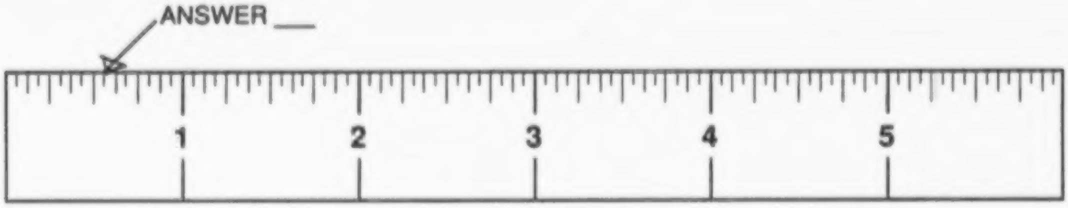
DO SKILL SHEET 5

USING A RULE

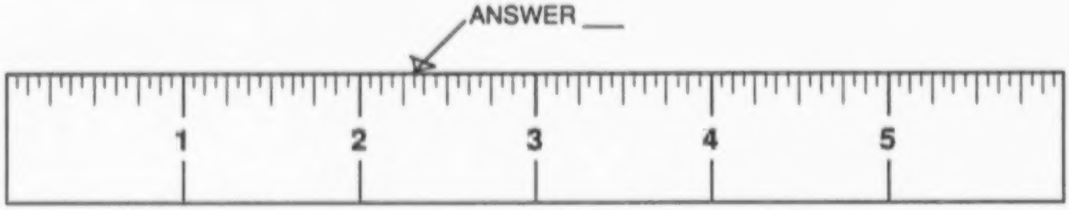
SKILL SHEET 5

Try the following measurements by yourself. Check with your instructor. If the fraction can be reduced, you must reduce it. Otherwise, your answer will not be correct.

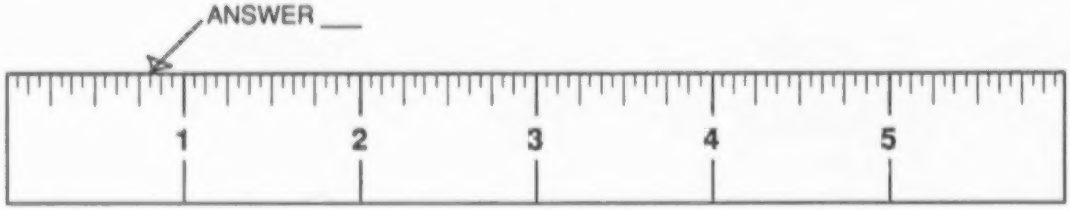
1.


ANSWER ___
1
2
3
4
5

2.


ANSWER ___
1
2
3
4
5

3.

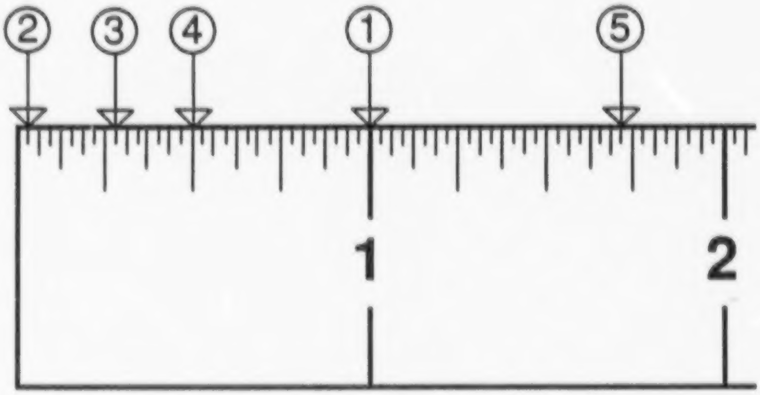

ANSWER ___
1
2
3
4
5

Another fine job!

5. 1/32nd-INCH SCALE

Now let's measure using 1/32nd of an inch. That's really small. The rule below is big so you can see the 32nd marks. Look at your rule too. These lines are tiny. You have to look hard to find them.

Look at the rule below. If you count the scales up to the one inch, you will find 32 marks. Each mark equals 1/32nd of an inch. Three marks would be 3/32nds; 21 marks would be 21/32nd. What would 22 marks be? That's right, 22/32nds. You could reduce that to 11/16ths.



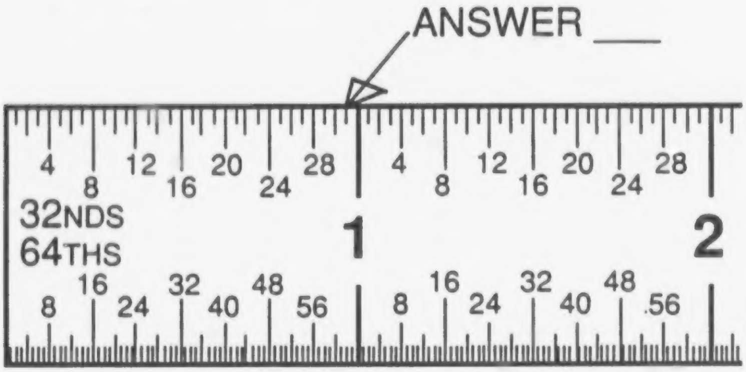
DO SKILL SHEET 6

USING A RULE

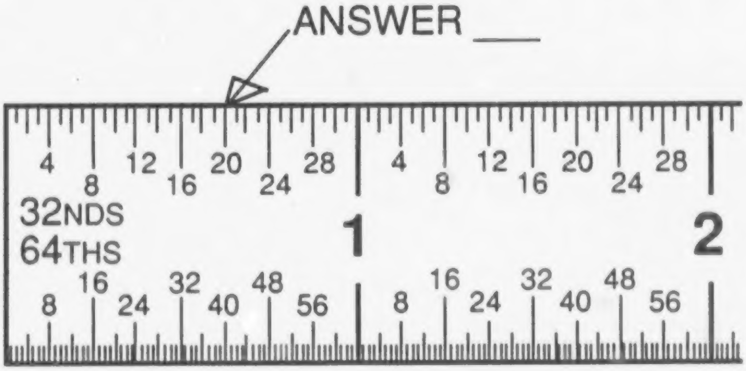
SKILL SHEET 6

Using the information you have, read the measurements. Check with your instructor.

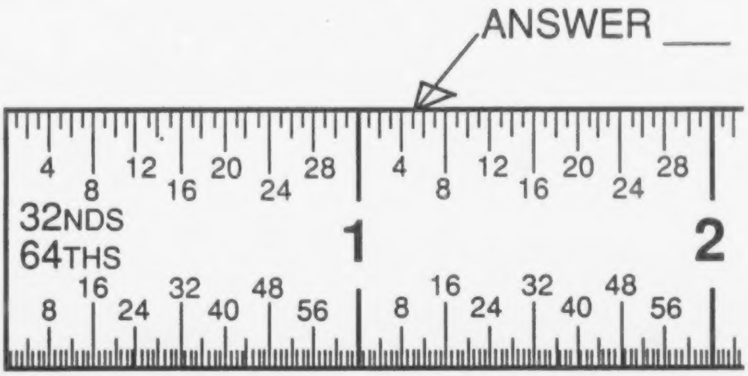
1.


ANSWER ___
4 8 12 16 20 24 28
32NDS
64THS
1
2
8 16 24 32 40 48 56

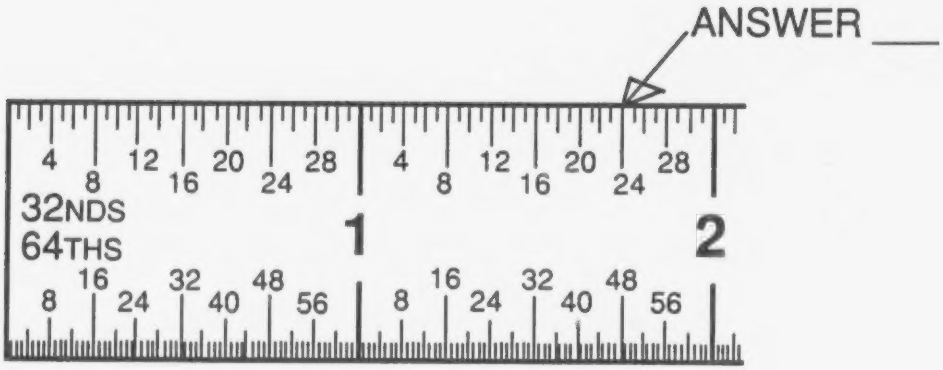
2.


ANSWER ___
4 8 12 16 20 24 28
32NDS
64THS
1
2
8 16 24 32 40 48 56

3.


ANSWER ___
4 8 12 16 20 24 28
32NDS
64THS
1
2
8 16 24 32 40 48 56

4.


ANSWER ___
4
8
12
16
20
24
28
32NDS
64THS
1
2
16
32
48
8
24
40
56

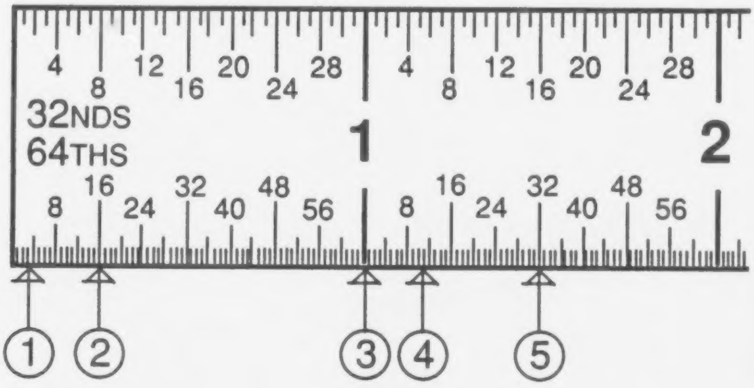
Guess what! You are nearing the end of this unit.

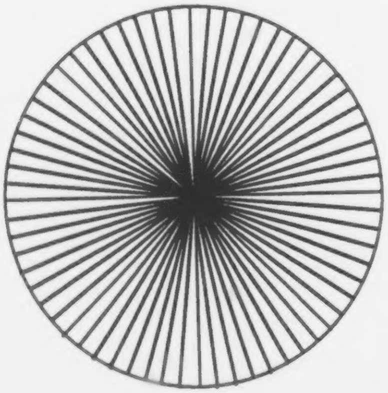
6. 1/64th-INCH SCALE

You now are ready to measure using the 1/64th-inch scale. When you divide the 32nds by two you get two 64th. The pie has enough pieces to feed a whole family reunion. The 1/64th-inch is the smallest division on the fractional steel rule.

Look at your rule. Can you find the 64th mark? It is the smallest mark on your rule. When you count from the end of the rule to the number 1, you will find there are 64 marks.

Look at the diagram, the arrow (↑) points that to 3/64ths. Again, there are times when you will need to reduce the fraction. If you can reduce a fraction and don't, your answer will be wrong.


4
8
12
16
20
24
28
4
8
12
16
20
24
28
32NDS
64THS
1
2
8
16
24
32
40
48
56
8
16
24
32
40
48
56
①
②
③
④
⑤



DO SKILL SHEETS 7 AND 8

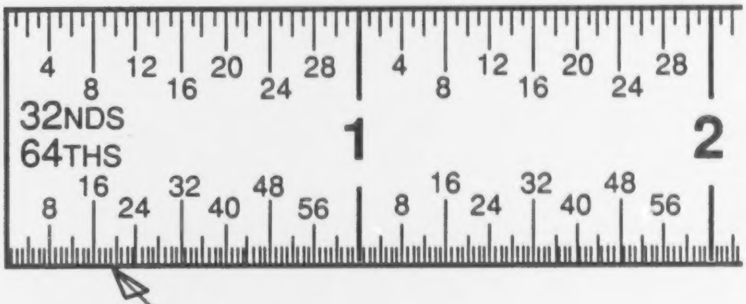
DO ACTIVITY SHEET 1

USING A RULE

SKILL SHEET 7

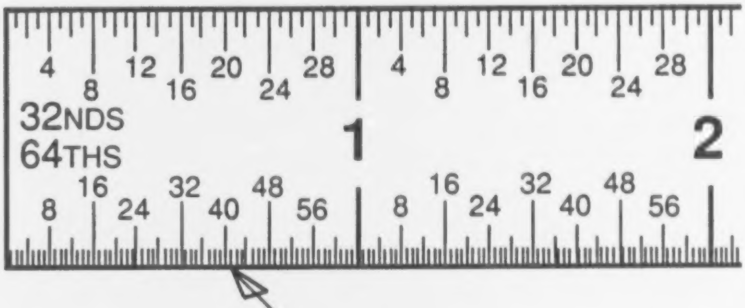
Let's try a few measurements. Write you answers on the space provided and then check with your instructor.

1.


4 8 12 16 20 24 28
32NDS
64THS
1
2
8 16 24 32 40 48 56

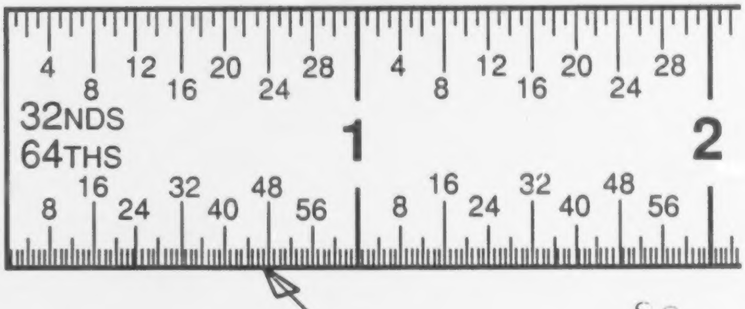
ANSWER ___

2.


4 8 12 16 20 24 28
32NDS
64THS
1
2
8 16 24 32 40 48 56

ANSWER ___

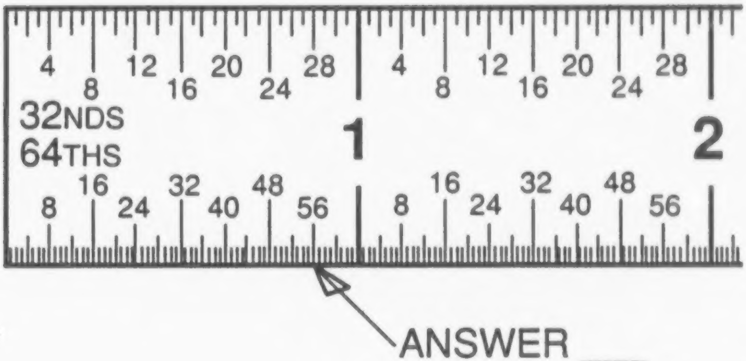
3.


4 8 12 16 20 24 28
32NDS
64THS
1
2
8 16 24 32 40 48 56

ANSWER ___

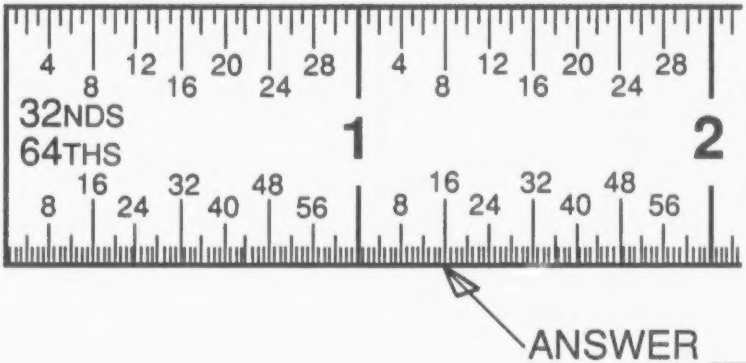
86

4.


4 8 12 16 20 24 28 4 8 12 16 20 24 28
32NDS
64THS
1
2
8 16 24 32 40 48 56 8 16 24 32 40 48 56

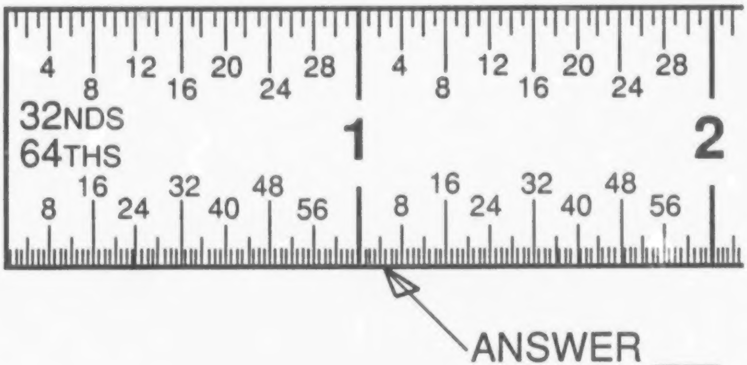
ANSWER ___

5.


4 8 12 16 20 24 28 4 8 12 16 20 24 28
32NDS
64THS
1
2
8 16 24 32 40 48 56 8 16 24 32 40 48 56

ANSWER ___

6.


4 8 12 16 20 24 28 4 8 12 16 20 24 28
32NDS
64THS
1
2
8 16 24 32 40 48 56 8 16 24 32 40 48 56

ANSWER ___

USING A RULE

SKILL SHEET 8

Fill in the blanks below to complete the sentences below. You will use the words in the list.

graduations rigid oil lay
cutters

1. Keep the rule clean, free of _________, chips or layout dye.

2. The rule is divided into measured scales or ___________ 1/2, 1/4, 1/8, etc.

3. Be careful how you use the rule, where and how you _______ it down.

4. Never bend a ___________ rule or over bend a flexible rule.

5. Keep the rule away from revolving ______________.

Mark "T" if the statement is true and "F" if the statement is false.

____ 6. Always use the rule for cleaning chips out of the machine or part.

____ 7. Be sure to stamp your initials or name on the rule.

____ 8. When measuring the 1 1/16th is a larger measurement than the 1/32nd.

____ 9. The steel rigid rule and the steel flexible rule are both used in measurement in the machine shop area.

____ 10. Place the rule on revolving cutters or parts.

USING A RULE

ACTIVITY SHEET 1

Using the workpieces your instructor provides, figure and record the measurements.

USING A RULE

UNIT TEST

Part 1. Match each number with a letter.

____ 1. will bend
____ 2. stiff, will not bend
____ 3. to bring into line
____ 4. measuring along a straight line
____ 5. marks showing different measured distances

a. flexible
b. rigid
c. linear measure
d. align
e. graduations

Part 2. Directions: Using the workpieces your instructor provides, figure and record the measurements.

USING A RULE

PERFORMANCE CHECKLIST

Student____________________________has successfully performed the following tasks:

	YES	NO	COMMENTS
1. Described care of the steel rule.			
2. Defined the terms used for the steel rule.			
3. Identified the various ways to measure with the various rules.			
4. Made measurements to 1/64th of an inch.			

Satisfactory - Should move on ☐

Repeat this unit ☐

____
Student's Signature/Date

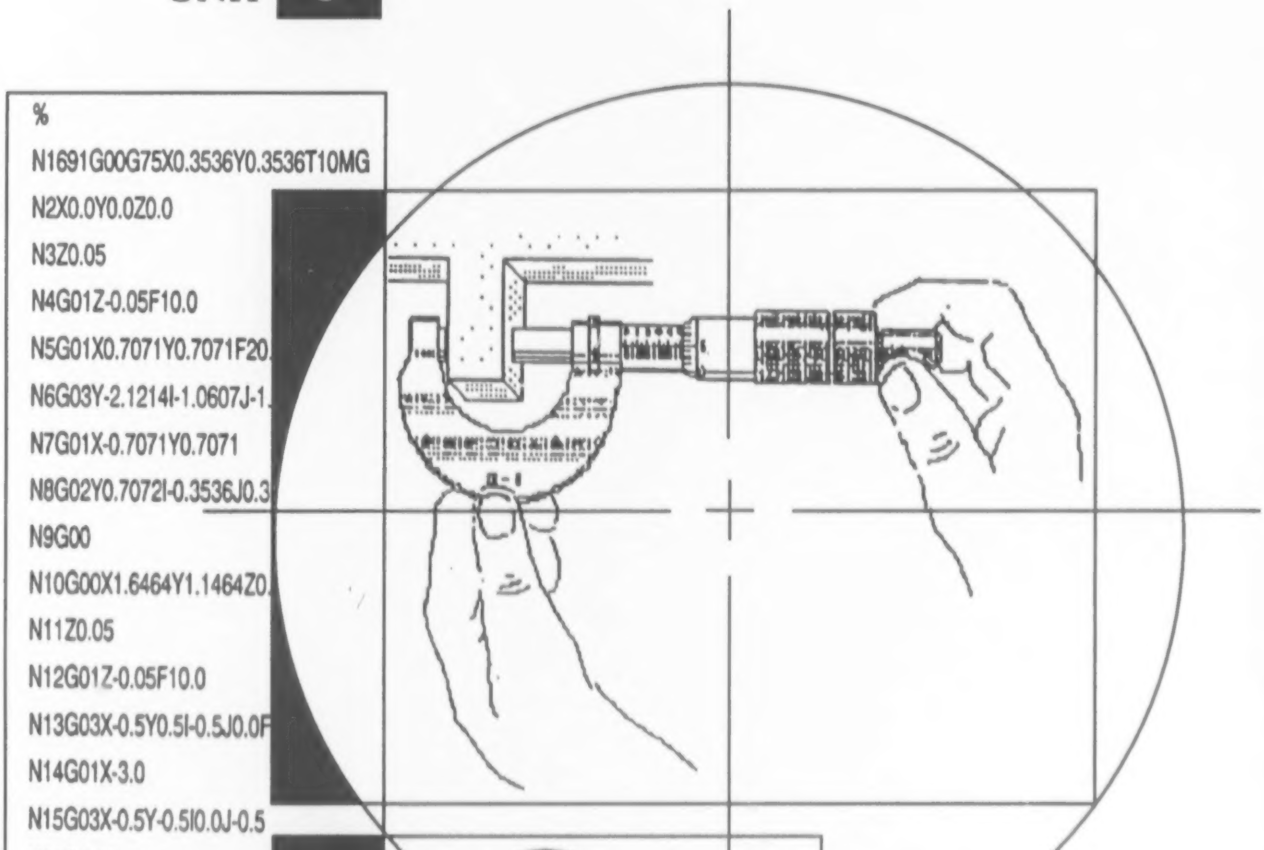
____
Instructor's Signature/Date

Machine Shop

90-9200-S
SUPPLEMENTARY UNITS

MICROMETERS

UNIT 3



```
%
N1691G00G75X0.3536Y0.3536T10MG
N2X0.0Y0.0Z0.0
N3Z0.05
N4G01Z-0.05F10.0
N5G01X0.7071Y0.7071F20.
N6G03Y-2.1214I-1.0607J-1.
N7G01X-0.7071Y0.7071
N8G02Y0.7072I-0.3536J0.3
N9G00
N10G00X1.6464Y1.1464Z0.
N11Z0.05
N12G01Z-0.05F10.0
N13G03X-0.5Y0.5I-0.5J0.0F
N14G01X-3.0
N15G03X-0.5Y-0.5I0.0J-0.5
N20G01Y3.0
N21G00
N22G90G00X-2.0y-1.5M2
```

IMI

MICROMETERS

INTRODUCTION

As a skilled metal worker, you must be able to make measurements exactly and correctly. A micrometer is an important tool for taking exact measurements. You must understand and be able to use a micrometer correctly to be successful in the machine shop.

UNIT OBJECTIVE

After completing this unit, you should be able to use and care for a micrometer correctly.

SPECIFIC OBJECTIVES

After completing this unit, you should be able to:

1. Select the proper size micrometer for each job.
2. Identify parts of a micrometer.
3. List rules for caring for micrometer.
4. Identify terms used when measuring with a micrometer.
5. Measure accurately with the micrometer.

Put a check beside the statement that is true for you.

☐ **I can do these tasks and want to go through the Performance Checklist.**

SEE YOUR INSTRUCTOR

☐ **I want to study the information in this unit before I do the Performance Checklist.**

TURN PAGE AND BEGIN

A. TERMS

1. **Measurement** - to find how long, thick, wide, round, flat, or curved an object is. To find how much an object will hold. To find the angle of an object.
2. **Frame** - something that makes a shape.
3. **Anvil** - the part of the micrometer that rests against an object.
4. **Spindle** - a rod that turns and is adjustable.
5. **Lock** - part of the micrometer that keeps it from moving from its setting.
6. **Sleeve** - a round part made to fit inside another round part on the micrometer.
7. **Thimble** - a round part made to fit over another round part and turn around it on the micrometer.
8. **Ratchet Stop** - a part of the micrometer used to adjust the micrometer to the proper feel when taking a measurement.
9. **Contact** - to touch against an object.
10. **Rotate** - to turn around.
11. **Standard (STD)** - a part that does not change in size or shape and is used to check a micrometer.
12. **Engraved** - a line of numbers cut into the metal parts of a micrometer.

13. **Horizontal** - a line that runs level to your eyesight.

14. **Vertical** - a line running up or down to a horizontal line.

15. **Graduation** - a mark on the micrometer that shows the amount of a measurement.

16. **Instrument** - the general word for precision tools.

17. **Reference Point** - the place on the micrometer where you read the measurement.

18. **Modified** - changed in some way.

19. **Interchangeable** - parts of a tool that can be exchanged with each other.

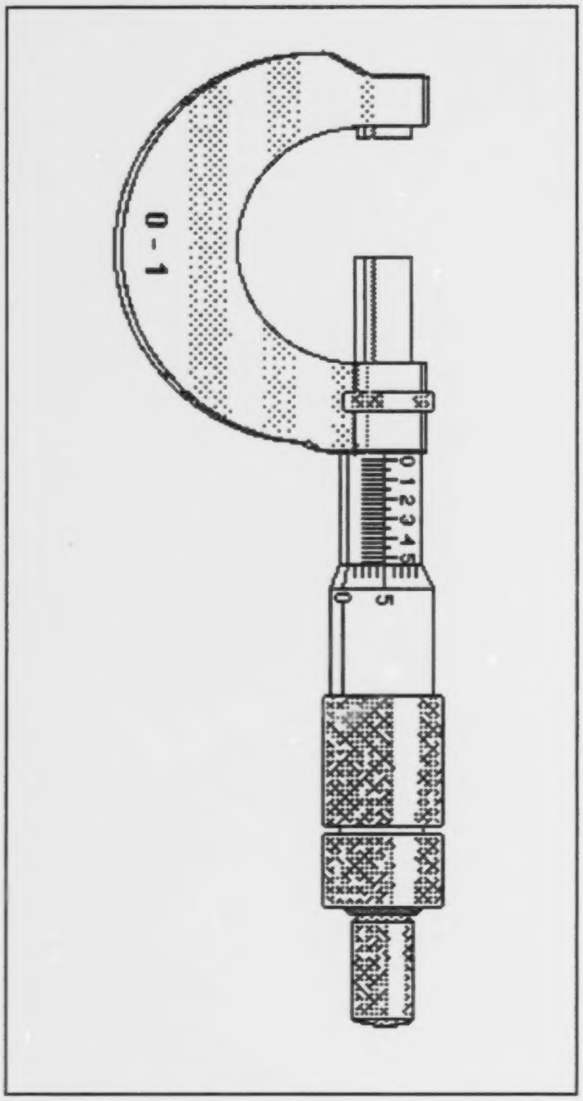
20. **Extension** - added to make longer.

21. **Division** - marks or numbers that divide a measurement into parts.

22. **Increment** - a number to be added.

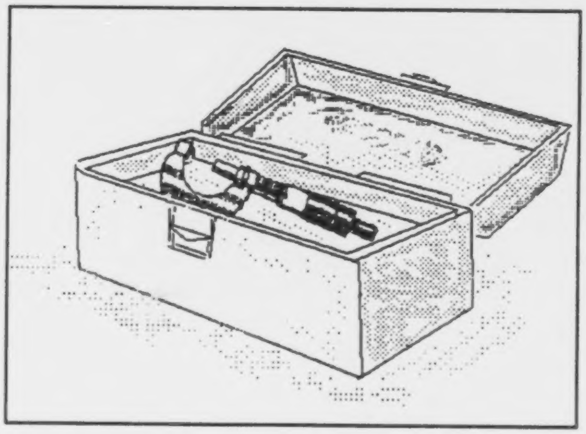
B. EQUIPMENT

Micrometer - A direct-reading caliper for measuring dimensions. A micrometer operates by using an accurate screw that may be turned in a fixed nut to vary the opening between the measuring faces.

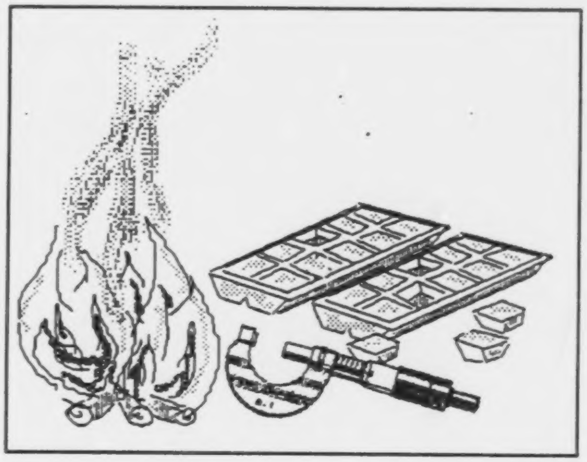


C. CARING FOR A MICROMETER

1. Keep the micrometer in a protected place where it can not come in contact with emery dust or chips.

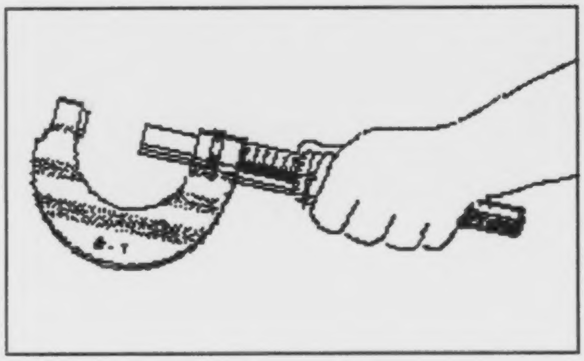


2. Avoid placing the micrometer where it will be affected by heat or cold. It can get bigger or smaller and will give a reading that is incorrect.

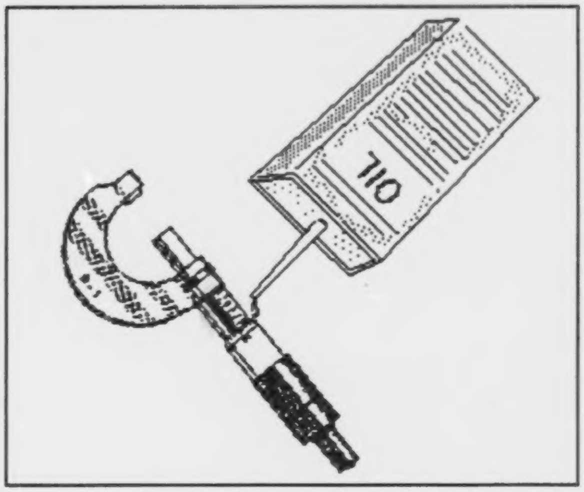


3. Do not place the micrometer on machinery that is operating. The micrometer could fall and be damaged.

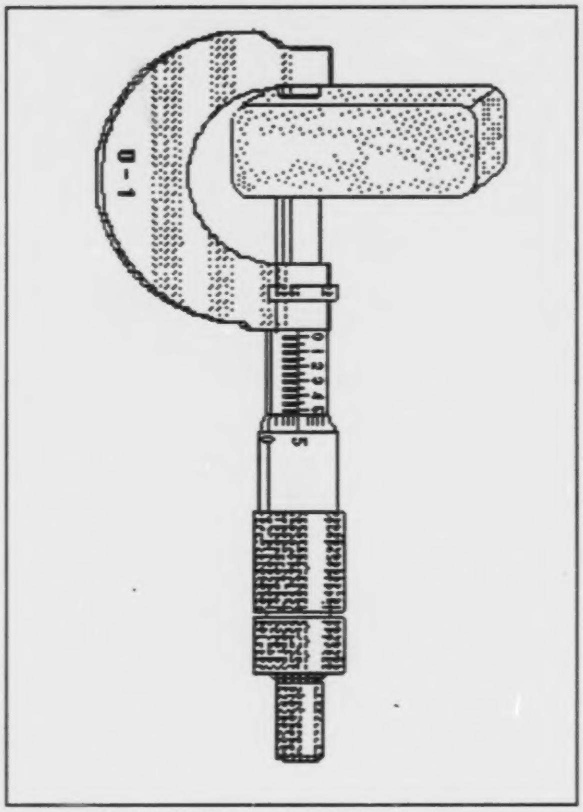
4. Do not drop the micrometer on the floor. Damage caused by dropping it may cause incorrect readings.



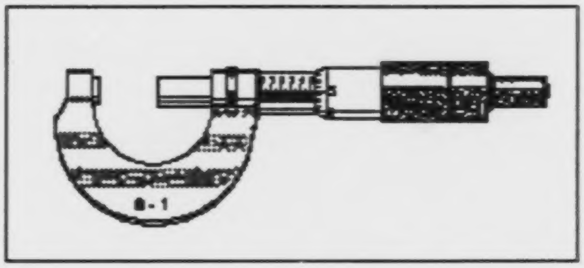
5. Oil the micrometer once in a while with a very light grade of oil to prevent rust and corrosion. One drop will be enough for the entire micrometer.



D. USING THE MICROMETER



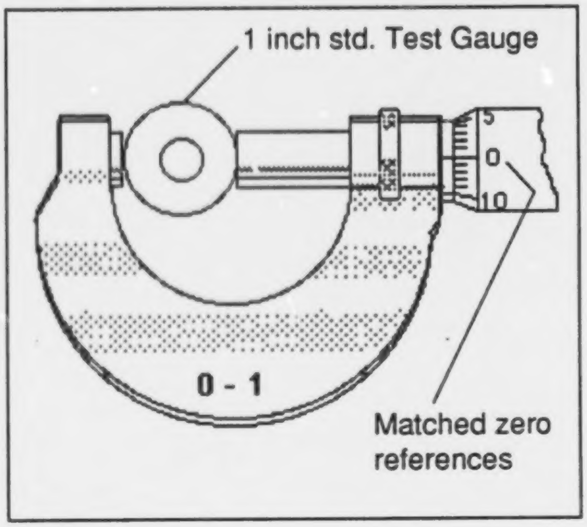
1. When you measure, bring the measuring surfaces into contact with the micrometer parts slowly and with a light pressure. This will protect the measuring surfaces from marks. It will also prevent springing the frame or bending the spindle.



2. When the micrometer is not in use, the spindle and anvil should not make contact.

3. Before you use a micrometer, be sure it is set on zero. This will assure accuracy.

4. Always clean faces of the anvil and spindle before each use.


1 inch std. Test Gauge
5
0
10
0 - 1
Matched zero references

5. Always rotate the thimble by using the slip clutch or ratchet, depending on which one your micrometer has.

6. Keep the micrometer in its case when you're not using it.

DO SKILL SHEET 1

MICROMETERS

SKILL SHEET 1

Listed below are some dos and don'ts for taking care of a micrometer. Write "do" in front of statements that tell you things you should do. Write "don't" in front of statements that tell you things not to do.

______ 1. Protect the micrometer from emery dust or chips.

______ 2. Place the micrometer where it will be affected by very cold and very hot temperatures.

______ 3. Drop the micrometer on the floor occasionally.

______ 4. Oil the micrometer occasionally.

______ 5. Store the micrometer on machinery that is operating.

______ 6. Let the spindle and anvil make contact when the micrometer is not in use.

______ 7. Clean faces of the anvil and spindle before you use the micrometer.

______ 8. Be sure the micrometer is set on zero before you use it.

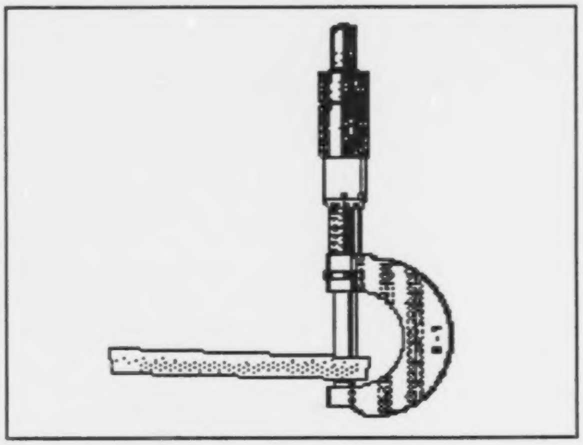
______ 9. Rotate the thimble by using a slip clutch or rachet stop.

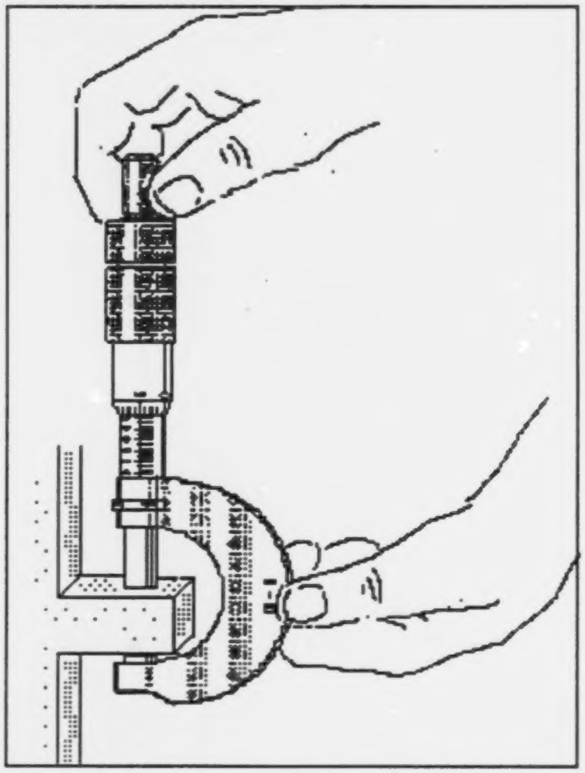
______10. Oil the micrometer occasionally to prevent rust and corrosion.

E. PARTS OF THE MICROMETER

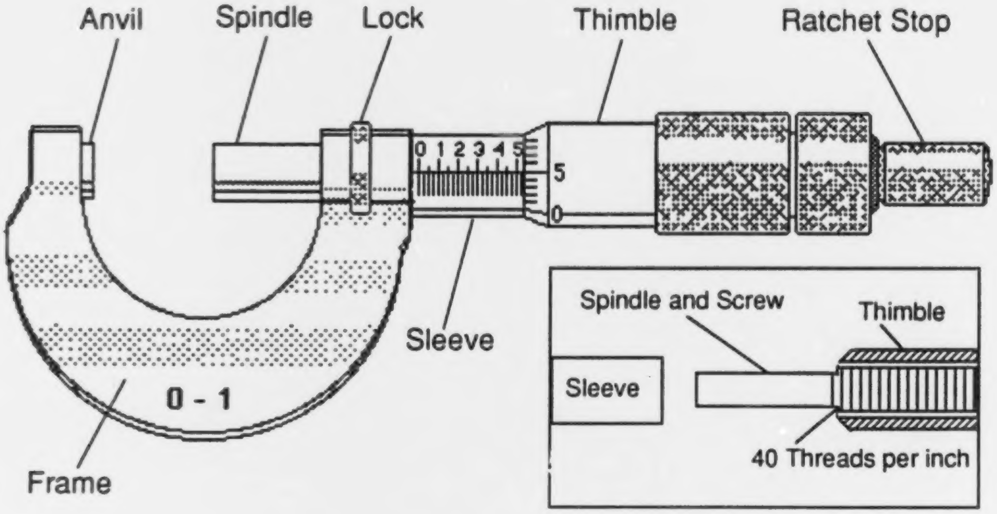
You need to ask yourself two main questions when you are deciding which micrometer to use: 1. What am I going to measure? 2. How will I measure this piece?

Micrometers come in different sizes and it is very important to use the correct size of micrometer when measuring items.





Look at the micrometer below. You should learn the names of its major parts. When you call the parts by their correct names, others will understand what you are talking about.

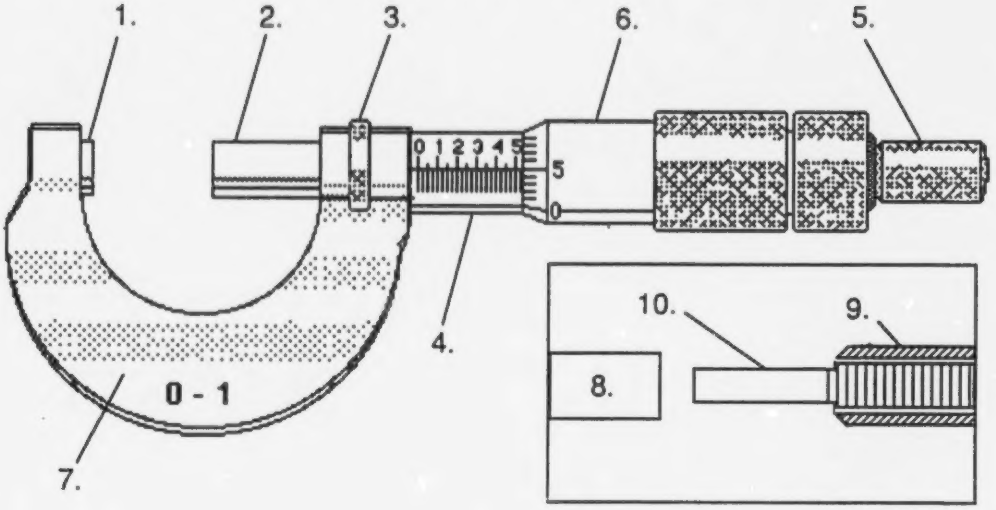

Anvil
Spindle
Lock
Thimble
Ratchet Stop
0 1 2 3 4 5
5
0
Sleeve
0 - 1
Frame
Spindle and Screw
Thimble
Sleeve
40 Threads per inch

DO SKILL SHEET 2

MICROMETERS

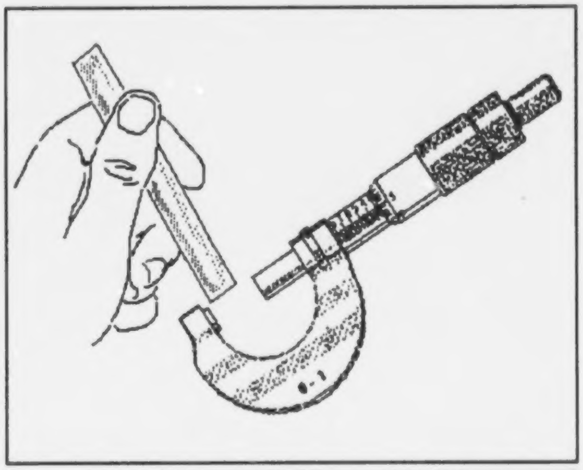
SKILL SHEET 2

Label the parts of the micrometer below.

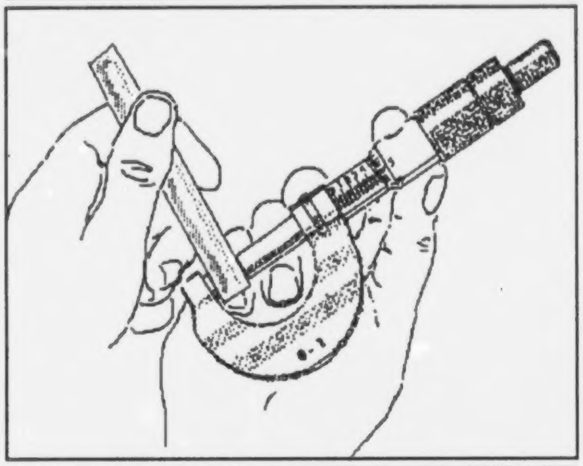

1.
2.
3.
6.
5.
0 1 2 3 4 5
5
0
4.
10.
9.
8.
0 - 1
7.

F. HOLDING THE MICROMETER

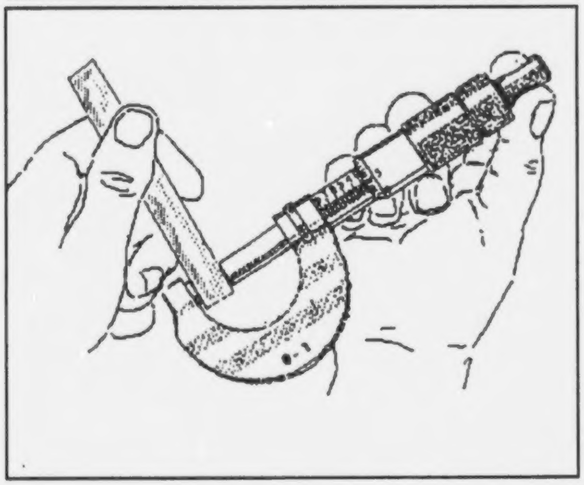
You must hold and use the micrometer properly. If you don't, the reading won't be accurate.



The first step is to open the micrometer slightly more than the diameter or thickness of the piece to be measured.



Second, slip the micrometer over the piece. Using your thumb and forefinger (the finger next to your thumb), turn the thimble to close the micrometer until it makes a slight pressure against the piece.



Third, secure the proper pressure of the spindle against the small stock measured by turning the knurled knob with the thumb and forefinger. Practice this also. It is important to be able to do this correctly when measuring the piece. As a protection, the ratchet or slip clutch built into the thimble of better micrometers limit the pressure being applied.

Use both hands to hold the micrometer when you measure a large piece or stationary parts.

DO ACTIVITY SHEET 1

MICROMETERS

ACTIVITY SHEET 1

1. Use a micrometer to practice using the thumb and forefinger to turn the thimble. Handle the micrometer with care as it is a precision measuring instrument. Ask your teacher to check you.

2. Slip the micrometer over a workpiece your teacher provides for you. Using your thumb and forefinger (the finger next to your thumb), turn the thimble to close the micrometer until it makes a slight pressure against the piece.

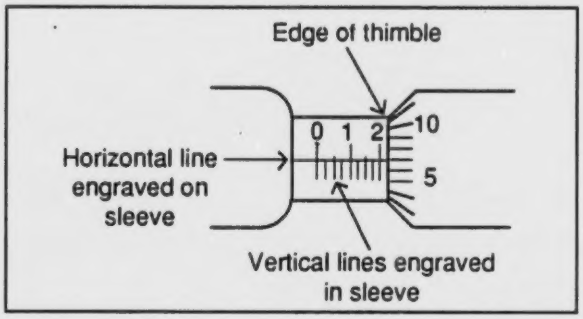
3. Secure the proper pressure of the spindle against a workpiece your teacher gives you to measure by turning the knurled knob also with the thumb and forefinger.

G. READING THE MICROMETER

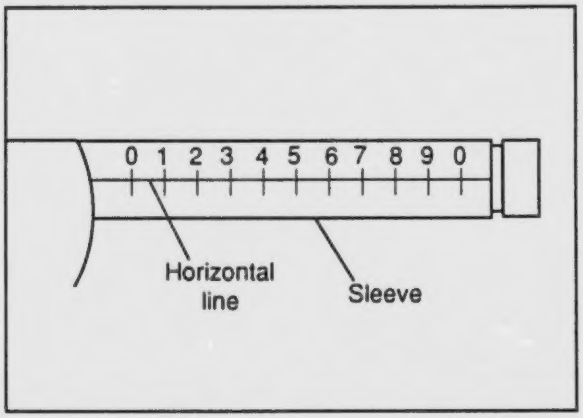
Let's first look at the micrometer before we do any actual measurement.

1. If you have a micrometer, start by unscrewing the thimble from the sleeve.

2. Set the thimble aside carefully.


Edge of thimble
0 1 2
10
5
Horizontal line engraved on sleeve
Vertical lines engraved in sleeve

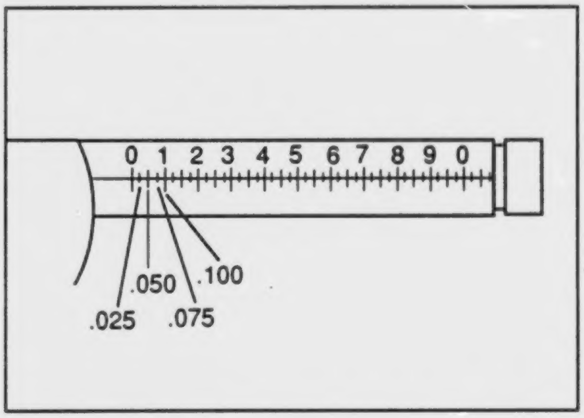
3. Look at the sleeve of the micrometer on the right. If you have micrometer look at its sleeve.


0 1 2 3 4 5 6 7 8 9 0
Horizontal line
Sleeve

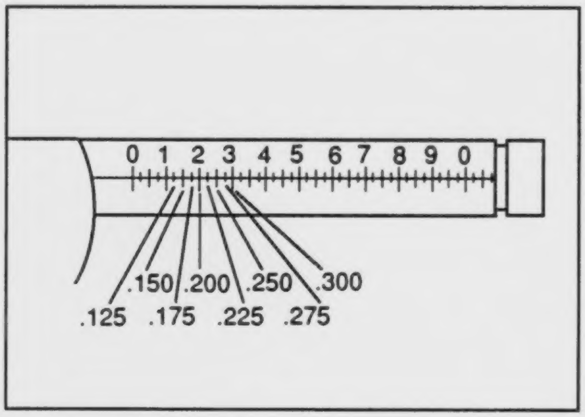
4. The sleeve will show you the values of the numbers engraved on it. These

numbers equal .100, .200, .300, etc. You read these as one hundred thousandths, two hundred thousandths, etc. These are called graduations.

5. Look at the vertical lines engraved on the sleeve. Each of these lines has a value of .025 (twenty-five thousandths). As you read each mark, add another .025 (twenty-five thousandths). The marks are .025, .050, .075 and .100.


0 1 2 3 4 5 6 7 8 9 0
.050
.100
.025
.075

6. After the numbers pass the .100, they start again. Look at the SLEEVE. The marks would read .125, .150, etc.


0 1 2 3 4 5 6 7 8 9 0
.150
.200
.250
.300
.125
.175
.225
.275

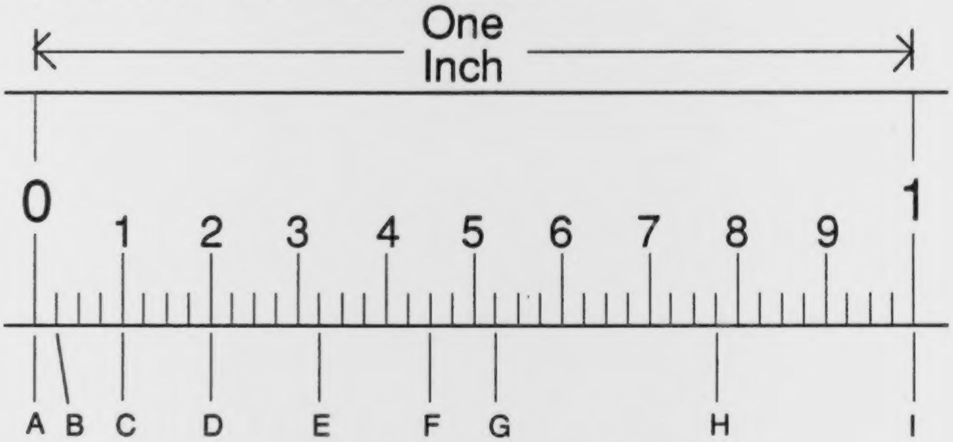
DO SKILL SHEET 3

MICROMETERS

SKILL SHEET 3

The arrow points to the marks to be read. Check with your instructor.

Write the answer in thousandths of an inch.


One Inch
0
1
2
3
4
5
6
7
8
9
1
A
B
C
D
E
F
G
H
I

A______

B______

C______

D______

E______

F______

G______

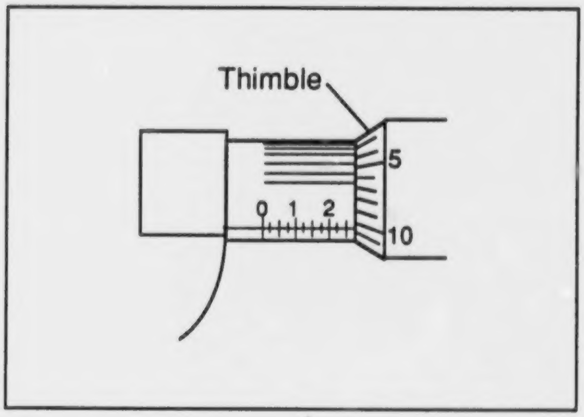
H______

I______

1 1 2

H. THE THIMBLE

The thimble is a very important part of the micrometer. Look at the diagram of the thimble.

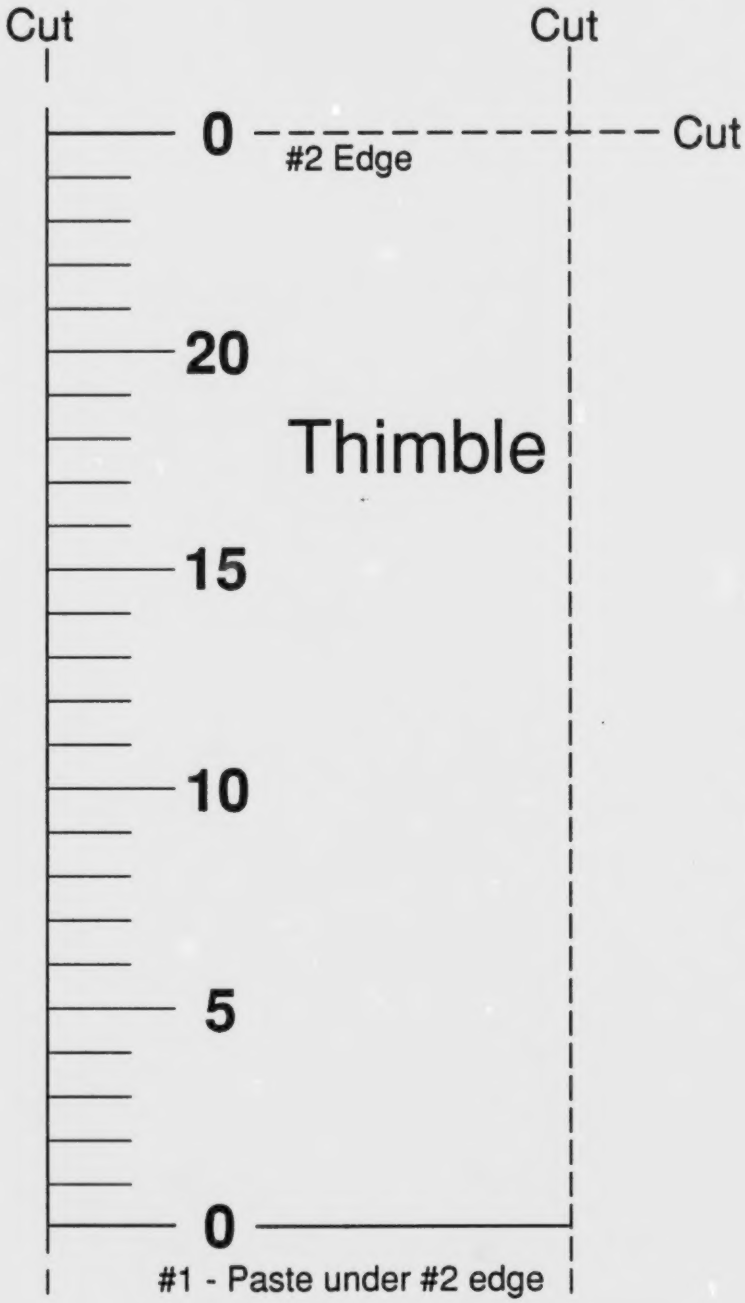


DO ACTIVITY SHEET 2

MICROMETERS

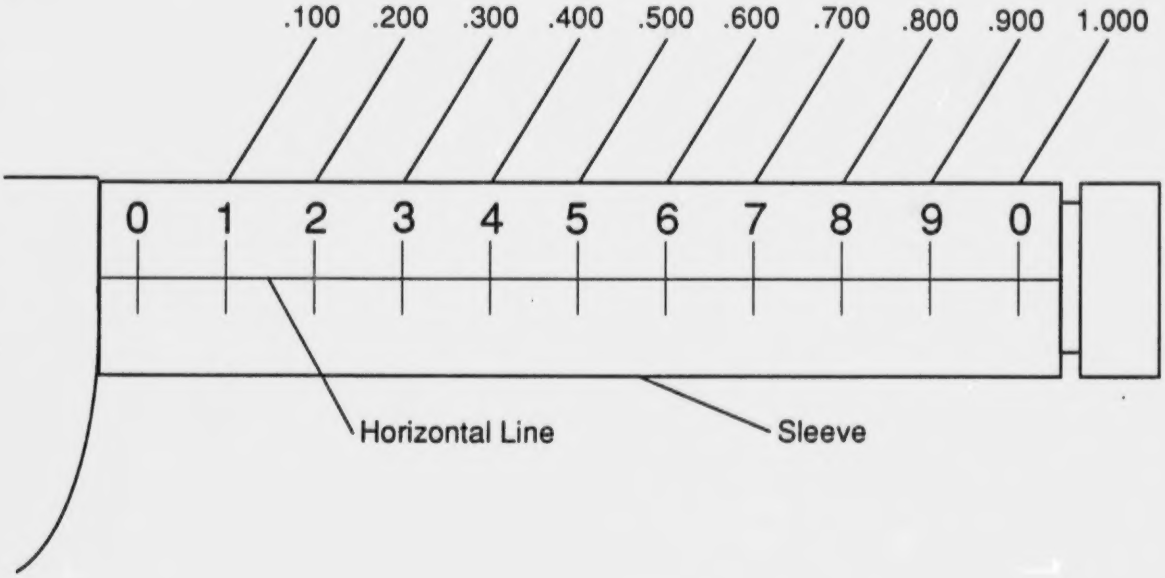
ACTIVITY SHEET 2

Cut out the thimble drawing on the next page. Cut along the dotted lines. Fasten the #1 edge under the #2 edge. When doing this, you should be able to cover all of the 0. You now have formed the thimble of the micrometer. Look at your model and look at the micrometer. See how they look alike. Look around the thimble and see how it is numbered 5, 10, 15, and 20, with the other numbers marked with a line. Each of these lines and numbers is equal to .001 (one thousandth). As each .001 is added to the one before, the value keeps increasing (.001, .002, .003, .004, .005, etc.)

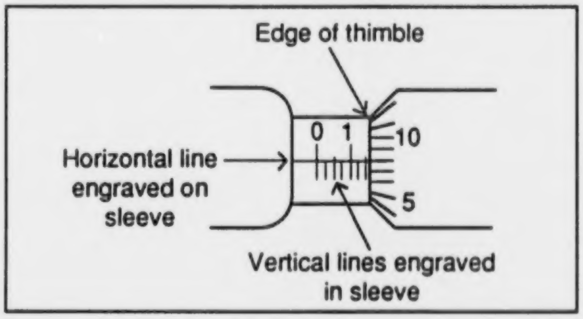

Cut
Cut
Cut
0
#2 Edge
20
Thimble
15
10
5
0
#1 - Paste under #2 edge

I. READING THE THIMBLE AND SLEEVE

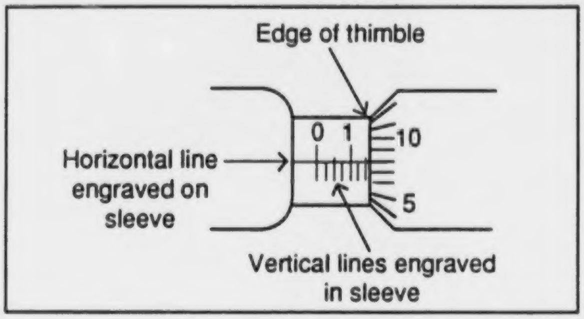
Look at the diagram. It shows how the numbers increase along the sleeve.


.100
.200
.300
.400
.500
.600
.700
.800
.900
1.000
0
1
2
3
4
5
6
7
8
9
0
Horizontal Line
Sleeve

Now look at this diagram. Check the edge of the thimble. The edge of the thimble is past the 1 on the sleeve. On a piece of paper write .100 (one hundred thousandths).


Edge of thimble
0
1
10
5
Horizontal line engraved on sleeve
Vertical lines engraved in sleeve

The edge of the thimble is also past 2 full spaces on the sleeve. Each of these spaces has a value of .025 (twenty-five thousandths). Therefore, the value of the 2 full spaces is .050 (fifty thousandths). Add the .050 to the .100 you read on the thimble. The total is .150 (one hundred fifty thousandths).


Edge of thimble
0 1
10
Horizontal line engraved on sleeve
5
Vertical lines engraved in sleeve

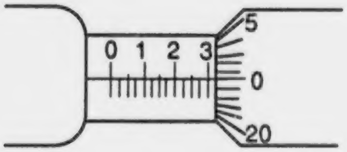
The edge of the thimble is a little past the .050 (fifty thousandths) mark, but not up to the 3rd vertical line. To find the value of this space, see which mark on the thimble lines up with the line on the sleeve. It is 8. It has a value of .008 (eight thousandths). Add the .008 to the total you have so far (.150). You now have .158 (one hundred fifty-eight thousandths), this is the final micrometer reading.

MICROMETERS

UNIT TEST

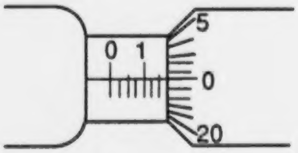
Part 1. Write each micrometer reading below.

1.



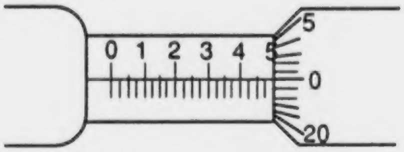
____

2.



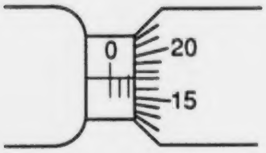
____

3.


0 1 2 3 4 5
5
0
20

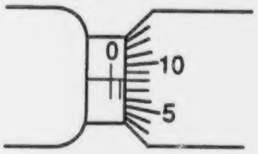
____

4.



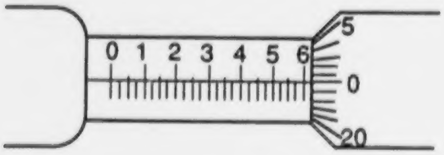
____

5.



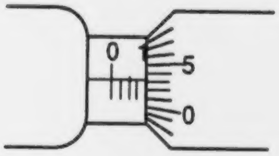
____

6.


0
1
2
3
4
5
6
5
0
20

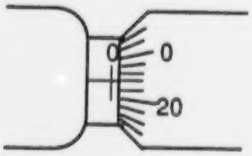
____

7.



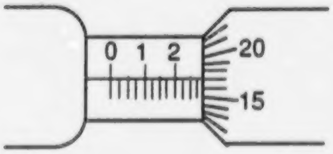
____

8.

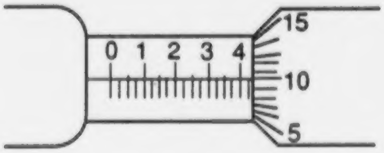


____

9.



10.


0
1
2
3
4
15
10
5

Part 2. Match the terms used with micrometers with their meanings.

A. increment
B. interchangeable
C. modified
D. instrument
E. vertical
F. engraved
G. rotate
H. division
I. extension
J. reference point
K. graduation
L. horizontal
M. Standard (STD)
N. contact

___11. To touch against an object.

___12. To turn around.

___13. A part that does not change in size or shape and is used to check a micrometer.

___14. A line of numbers cut into the metal parts of a micrometer.

___15. A line that runs level to your eyesight.

___16. A line running up or down to a horizontal line.

___17. A mark on the micrometer that shows the amount of a measurement.

___18. The general word for precision tools.

___19. The place on the micrometer where you read the measurement.

___20. Changed in some way.

___21. Parts of a tool can be exchanged with each other.

___22. Added to make longer.

___23. Marks or numbers that divide a measurement into parts.

___24. A number to be added.

MICROMETERS

PERFORMANCE CHECKLIST

Student ______________________________ has successfully performed the following tasks:

	YES	NO	COMMENTS
1. Selected the proper size micrometer for each job.			
2. Identified parts of a micrometer.			
3. Identified rules for caring for the micrometer.			
4. Identified terms used when measuring with a micrometer.			
5. Measured accurately with the micrometer.			

Satisfactory - Should move on ☐

Repeat this unit ☐

_________________________ Student's Signature/Date

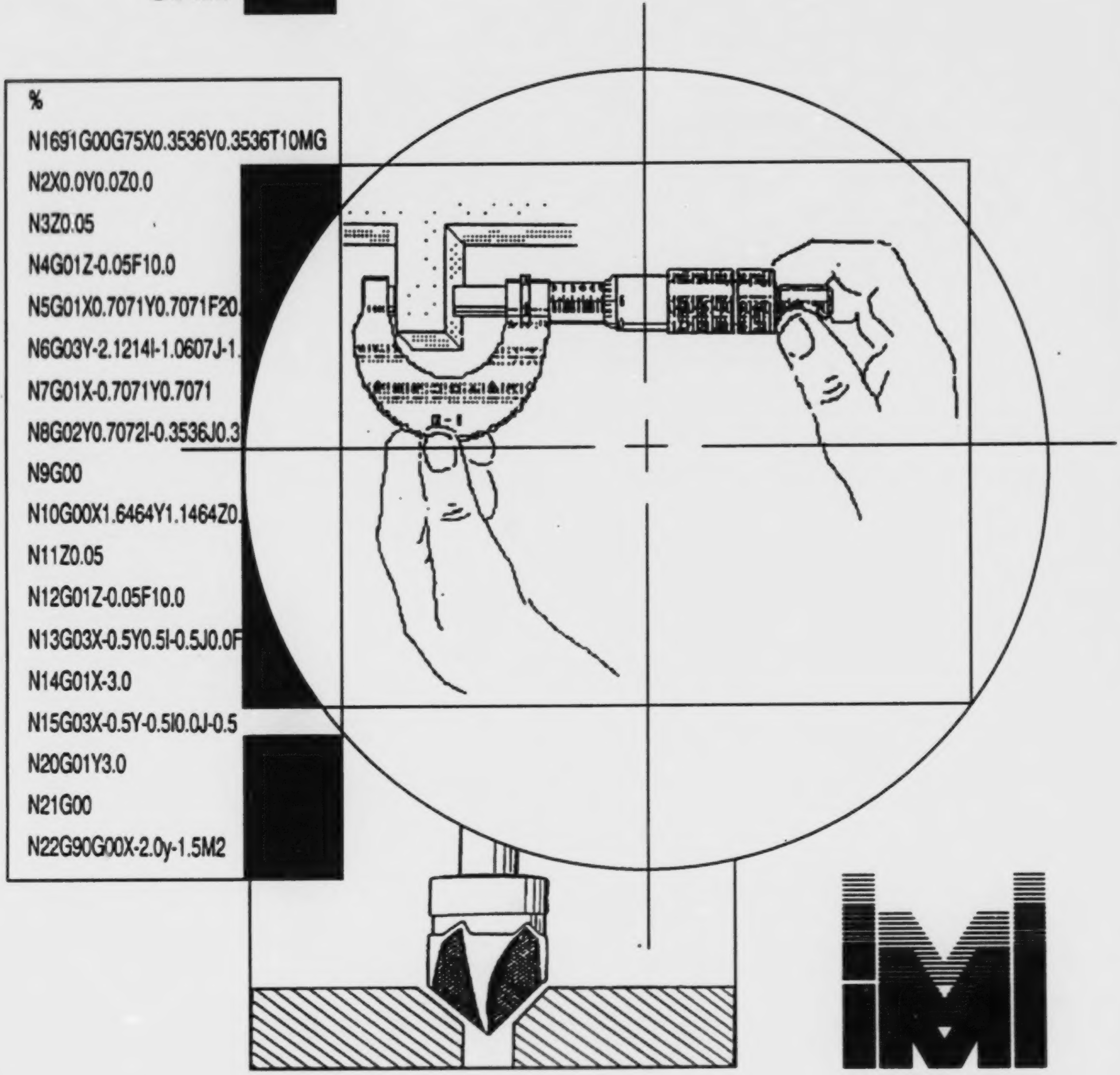
_________________________ Instructor's Signature/Date

Machine Shop

90-9200-S
SUPPLEMENTARY UNITS

BLADES FOR SAWING

UNIT 4



```
%
N1691G00G75X0.3536Y0.3536T10MG
N2X0.0Y0.0Z0.0
N3Z0.05
N4G01Z-0.05F10.0
N5G01X0.7071Y0.7071F20.
N6G03Y-2.1214I-1.0607J-1.
N7G01X-0.7071Y0.7071
N8G02Y0.7072I-0.3536J0.3
N9G00
N10G00X1.6464Y1.1464Z0.
N11Z0.05
N12G01Z-0.05F10.0
N13G03X-0.5Y0.5I-0.5J0.0F
N14G01X-3.0
N15G03X-0.5Y-0.5I0.0J-0.5
N20G01Y3.0
N21G00
N22G90G00X-2.0y-1.5M2
```

IMI

BLADES FOR SAWING

INTRODUCTION

Selecting the right blade for sawing each job is important. If you choose the correct blade, the work will go faster and the saw blade will last longer.

UNIT OBJECTIVE

After completing this unit, you should be able to choose the correct blade for power hacksaw and bandsaw operation and for the materials being used.

SPECIFIC OBJECTIVES

After completing this unit, you should be able to:

1. Identify important characteristics of saw blades.
2. Identify characteristics of a workpiece that affect sawing.
3. Identify factors in selecting blades for power hacksaws.
4. Select blades for power hacksawing.
5. Identify factors in selecting blades for bandsawing and contouring.
6. Select blades for bandsawing and contouring.

Put a check beside the statement that is true for you.

☐ I can do these tasks and want to go through the Performance Checklist.

SEE YOUR INSTRUCTOR

☐ I want to study the information in this unit before I do the Performance Checklist.

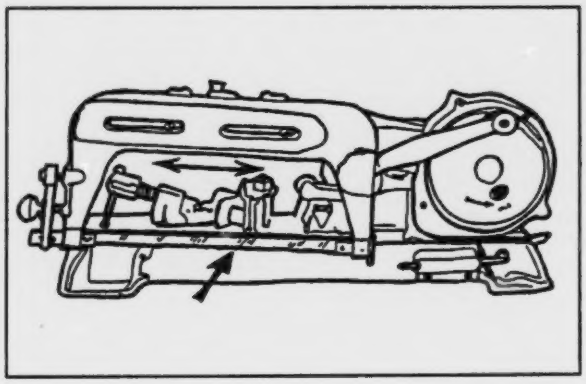
TURN PAGE AND BEGIN

A. TERMS

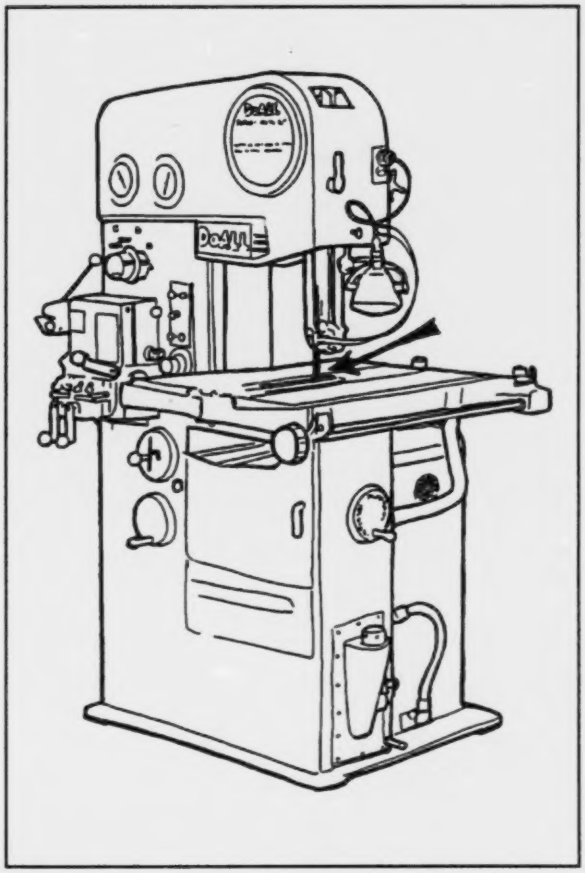
1. **Blade materials** - the type of metal from which the blade is made. Saw blades for power hacksaws and bandsaws are made from carbon steels and high-speed alloy steels. Blades may also have tungsten carbide tipped teeth.
2. **Blade kerf** - the kerf of a saw cut is the width of the cut the blade produces.
3. **Blade width** - the width of a saw blade is the distance from the tip of the tooth to the back of the blade.
4. **Blade pitch** - the pitch of a saw blade is the number of teeth per inch. An eight-pitch blade has eight teeth per inch (a tooth spacing of 1/8 inch).
5. **Tooth forms** - tooth form is the shape of the saw tooth. Saw tooth forms are either standard, skip, or hook.
6. **Set** - the offset of the teeth on each side to provide clearance for the back of the blade.
7. **Set patterns** - set forms include raker, straight, and wavy. Raker and wavy are the most common.
8. **Reciprocating** - to move back and forth like the action of a power hacksaw.

B. EQUIPMENT

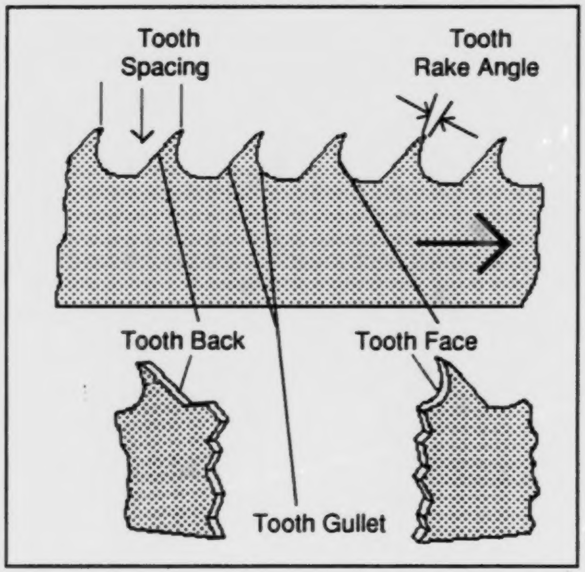
Power hacksaw blade - a thick, flat piece of metal with teeth on one edge used for cutting.



Bandsaw blade - circular band of metal with teeth on one edge used for cutting.



C. BLADE CHARACTERISTICS


Tooth Spacing
Tooth Rake Angle
Tooth Back
Tooth Face
Tooth Gullet

When you talk about saw blades, you must be familiar with certain terms. The picture at the right shows you the parts of a saw blade

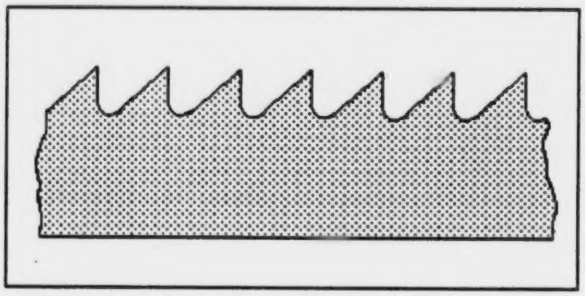
You must consider five characteristics each time you select a saw blade. They are material, tooth form, pitch, width and set.

1. **Material.** Manufacturers produce saw blades from several materials. These include: carbon steel, high-speed steel and other alloy steels. There are also composite blades that consist of high-speed or

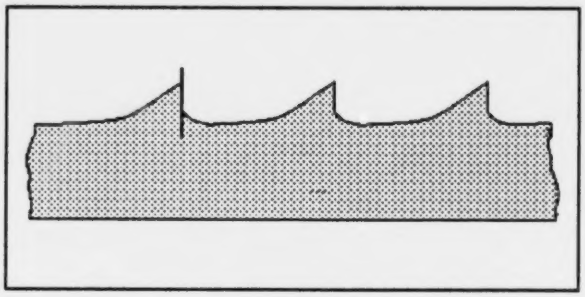
tungsten carbide teeth welded to a carbon steel back.

2. **Tooth form** refers to the shape of the teeth. Common tooth forms are regular, skip or hook.

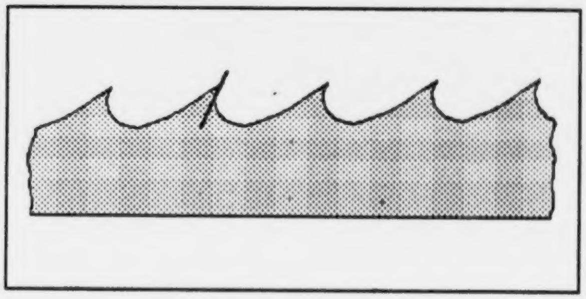
   a. **Regular or standard** tooth forms are used with all blades having a pitch of six or more teeth per inch.



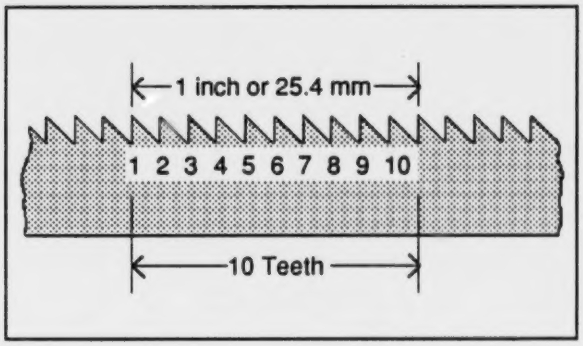
   b. **Skip tooth** or buttress form has the same shape as regular form but has more space between the teeth. This provides more chip clearance when you cut soft materials.



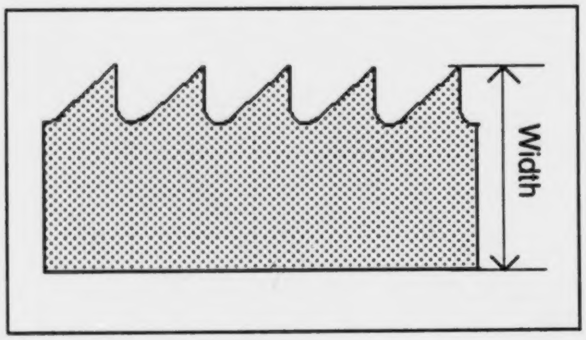
   c. **Hook tooth** blades are used for blade pitch of six less teeth per inch.



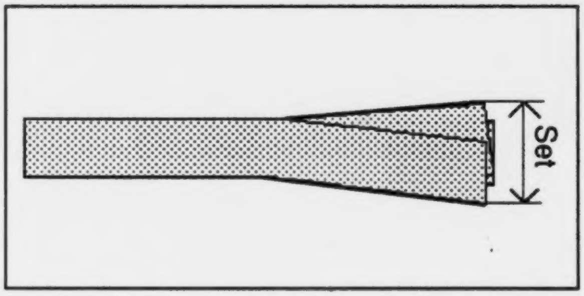
3. **Pitch** is the number of teeth per inch of the blade. The thickness of the material you are cutting will determine the pitch you need. Blades come in pitches from 2 to 32. An eight-pitch blade has eight teeth per inch (a tooth spacing of 1/8 inch).



4. **Width** is the distance from the back of the blade to the top of the teeth. Width is an important consideration in contour sawing because a blade that is too wide cannot saw around tight corners.

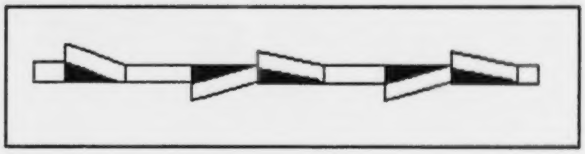


5. **Set** means the way the teeth are bent alternately from left to right. Offsetting the teeth makes room for the blade

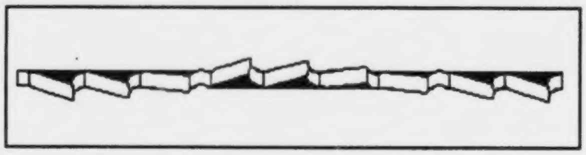


itself during cutting. There are three basic set patterns:

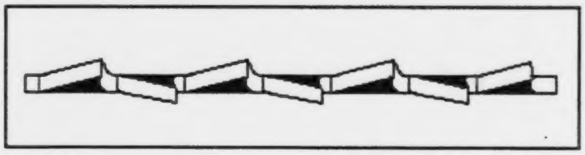
a. **Raker** has one tooth to the left, one centered and one set to the right. The raker set is used for most general purpose and contour cutting.



b. **Wavy** set weaves back and forth from left to right. It is especially used for blades with a fine pitch.



c. **Straight** set alternates each tooth from left to right. It is seldom used in metal working.



DO SKILL SHEET 1

DO ACTIVITY SHEET 1

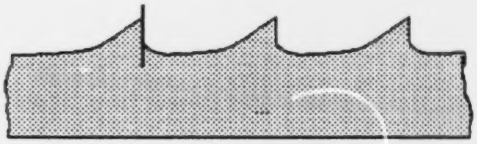
BLADES FOR SAWING

SKILL SHEET 1

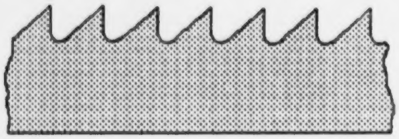
Part 1. Write the correct name of the saw tooth form under each picture. You will use these words:

Standard Skip Hook

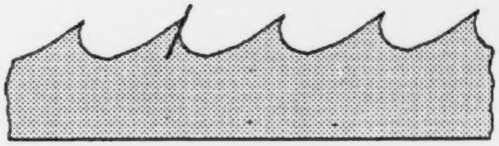
1.



2.

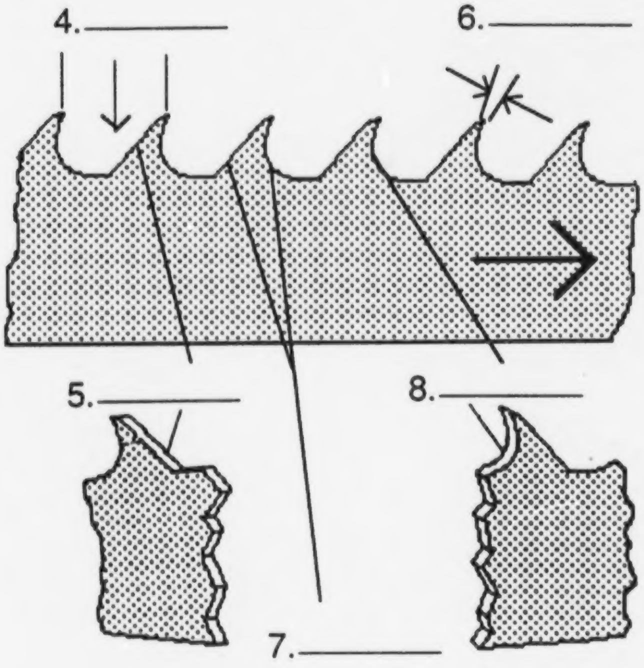


3.



Part 2. Label the parts of the saw blade below. You will use these words:

Tooth spacing	Tooth guliet	Tooth back
Tooth face	Rake angle	



Part 3. Match four saw blade characteristics with their correct definitions.

A. Tooth form
B. Pitch
C. Set
D. Width

___ 9. Offset of the teeth from side to side to give clearance for the band in the saw kerf or cut.

___10. The number of teeth per inch.

___11. The shape of the teeth: standard, skip or hooked.

___12. The distance from the tip of the tooth to the back of the blade.

BLADES FOR SAWING

ACTIVITY SHEET 1

Your instructor will provide you with several saw blades. Identify the following characteristics for each one.

A. Set

B. Estimate of width of kerf the blade will produce (measure across the set)

C. Pitch

D. Width

E. Type (band or reciprocating)

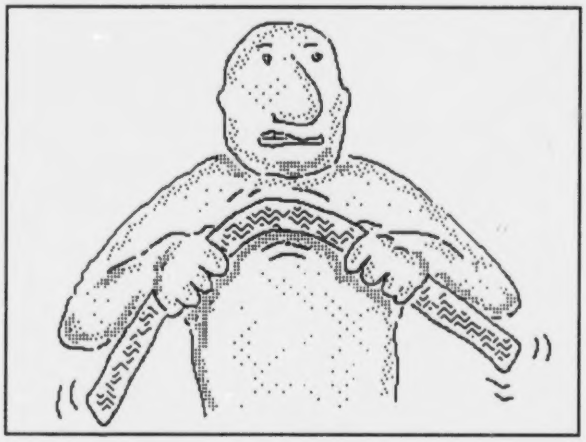
D. WORKPIECE CHARACTERISTICS

As a machinist, you must learn to cut all kinds of materials. You will work with many types. You must be able to identify some important characteristics of the material you will be cutting before you can choose the best blade for the job.

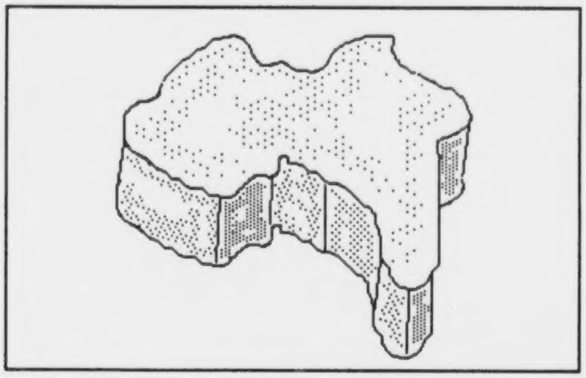
1. **Material.** Common workpiece materials that you may saw in the shop include: steels, aluminums, brass and other copper alloys, plastics and woods.

2. **Hardness** of the material determines how much force you must apply to cut it. Also, the blade must be made of harder metal than

the material you are cutting. For cutting hard materials such as steels, you must use a carbide or high-speed steel blade. You can cut soft materials such as aluminum with a high-speed steel blade.



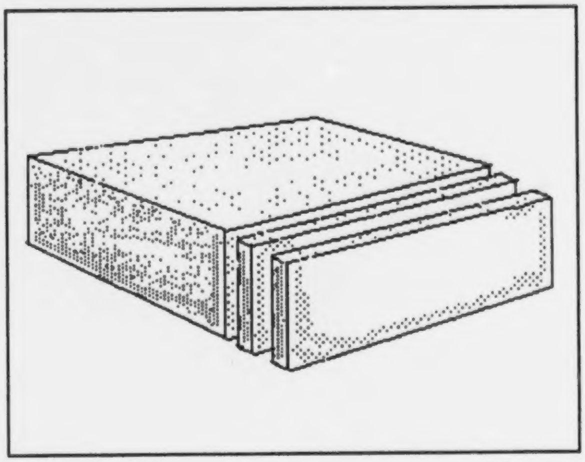
3. The **ductility** of a material (its ability to be stretched without breaking) is important. Very soft, ductile materials form a large chip when you machine them. A skip tooth blade works best with soft metals such as copper, aluminum, etc.



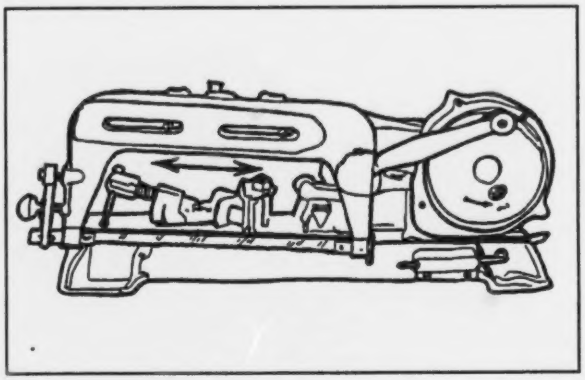
4. **Shape** is important in contour sawing. A wide blade will not let you cut

small curves and corners.

5. **Cross-section**. When you are cutting a part with a thin cross-section, you must be sure your blade has a sufficient number of teeth per inch (pitch). As a general rule, no fewer than two teeth should be in contact with the cross-section of the work at any time.



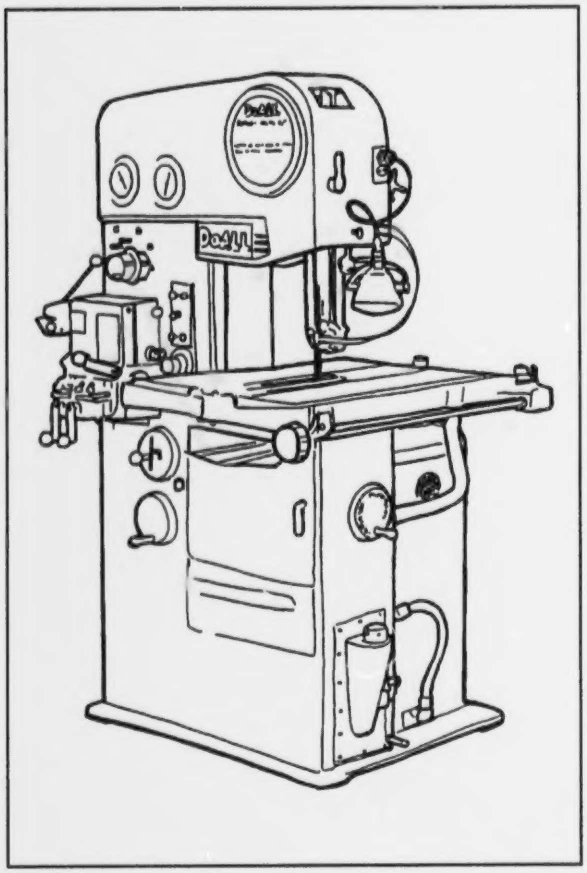
E. SELECTING A BLADE



1. **Power hacksaws.** Several factors affect your choice of a power hacksaw blade. Always consider the following:

   a. No fewer than two teeth must contact the workpiece during cutting.
   b. Soft materials require a coarse-tooth blade.
   c. Thin sections and hard materials require a finer pitch blade.
   d. Hard materials may require you to use a high alloy or tungsten carbide saw blade.

2. **Band machining.** Consider the following factors when you select a bandsaw:
   a. No fewer than two teeth must contact the workpiece during cutting.
   b. Soft materials require a



skip toothed band.

c. Thin sections and hard materials require a finer pitch band.

d. Use a narrower band when cutting contours than you would for straight cutting.

e. Hard materials may require that you use a high alloy band.

Often you can look at a chart to help you select the right saw blade. However, before you refer to a chart, you must know the type of material you will be sawing.

DO SKILL SHEET 2

DO ACTIVITY SHEET 2

BLADES FOR SAWING

SKILL SHEET 2

Choose the best response to each question.

___ 1. What is the least number of teeth that should touch the workpiece?

A. 1 C. 3
B. 2 D. 4

___ 2. Generally, what type (form) of blade teeth would you use to cut soft materials?

A. Standard C. Skip
B. Hook

___ 3. Which two workpiece characteristics would cause you to choose a fine-pitch blade?

A. Thin sections C. Soft materials
B. Thick sections D. Hard materials

___ 4. A narrow blade is best suited for which application?

A. Straight cutting
B. Contour sawing

BLADES FOR SAWING

ACTIVITY SHEET 2

Your instructor will provide you with a variety of saw blades (or a catalog of saw blades) and several workpieces. The workpieces will be made of different materials and contain cross-sections, angles, extrusions, etc. Decide which blade to use for each workpiece.

143

BLADES FOR SAWING

UNIT TEST

Part 1. Match the terms below with their definitions.

A. Blade kerf   B. Blade width   C. Blade pitch

D. Set

___ 1. Means the teeth are bent alternately from left to right to provide clearance for the blade during cutting

___ 2. Refers to the number of teeth per inch

___ 3. Distance from the back of the blade to the top of the teeth

___ 4. Refers to the cut produced by the saw blade

Part 2. Choose the best answer.

___ 5. A composite saw blade is made from which material?

A. carbon steel

B. high-speed steel

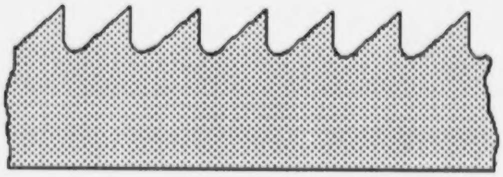
C. Alloy steels

D. high-speed steel welded to a carbon steel back

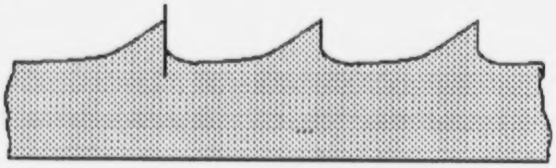
Part 3. Match these tooth forms with the correct illustration.

A. Regular B. Hook C. Skip

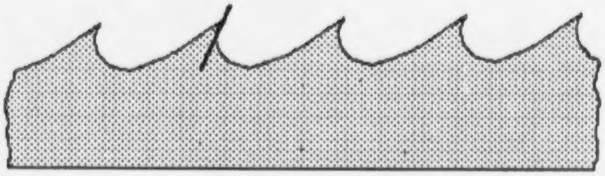
6.



7.



8.



Part 4. Which three properties or characteristics of workpiece materials can affect blade selection. (Give three answers)

___ 9.
A. Hardness
B. Ductility
C. Color
D. Temperature
E. Shape

Part 5. Choose the best response to each question.

___10. What is the least number of teeth that should touch the workpiece?

A. 1
B. 2
C. 3
D. 4

___11. Generally, what type (form) of blade teeth would you use to cut soft materials?

A. Standard
B. Hook
C. Skip

___12. Which two workpiece characteristics would cause you to choose a fine-pitch blade?

A. Thin sections
B. Thick sections
C. Soft materials
D. Hard materials

___13. A narrow blade is best suited for which application?

A. Straight cutting
B. Contour sawing

BLADES FOR SAWING

PERFORMANCE CHECKLIST

Student____________________________has successfully performed the following tasks:

	YES	NO	COMMENTS
1. Identified important characteristics of saw blades.			
2. Identified characteristics of a workpiece that affect sawing.			
3. Identified factors in selecting blades for power hacksaws.			
4. Selected blades for power hacksawing.			
5. Identified factors in selecting blades for bandsawing and contouring.			
6. Selected blades for bandsawing and contouring.			

Satisfactory - Should move on ☐
Repeat this unit ☐

____
Student's Signature/Date

____
Instructor's Signature/Date

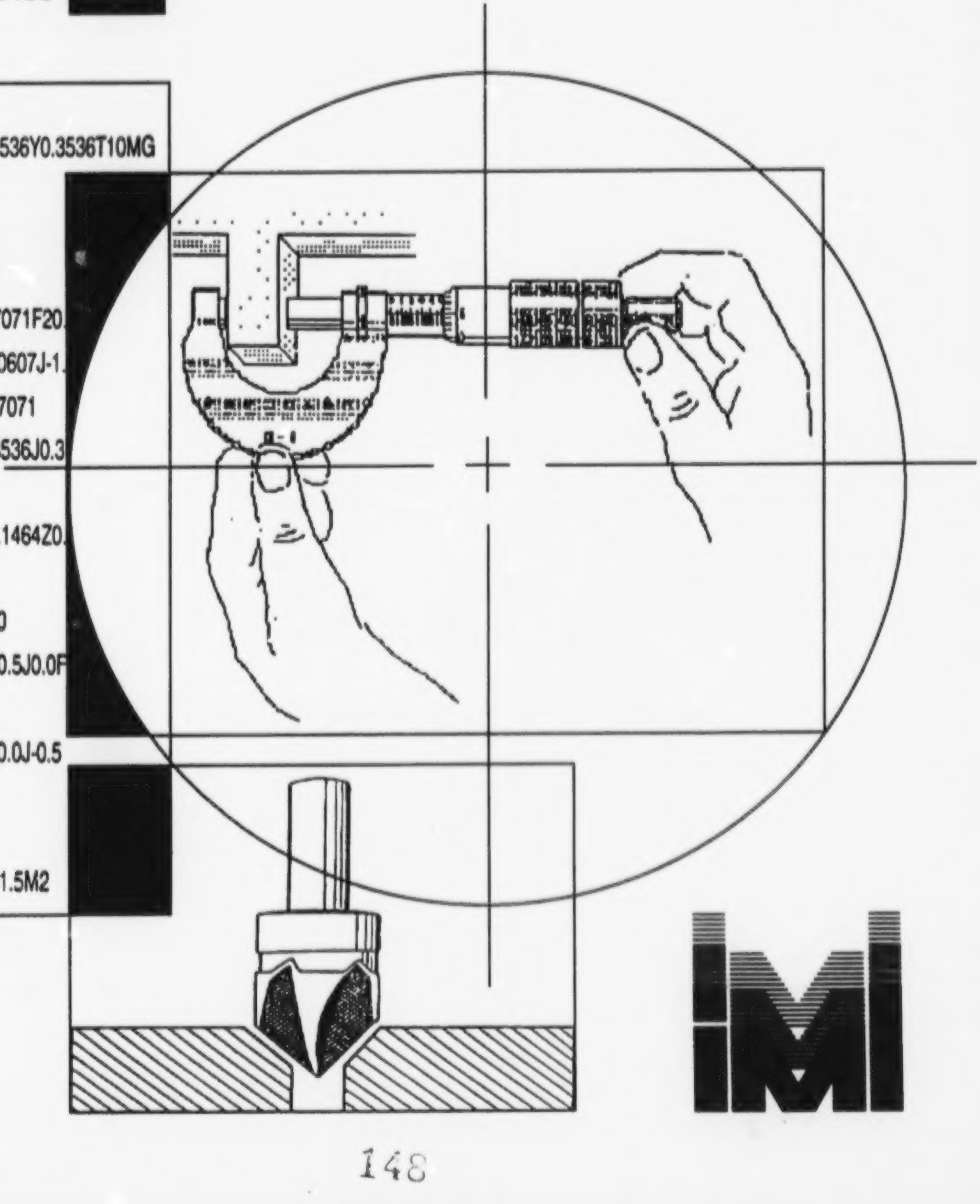
Machine Shop

90-9200-S
SUPPLEMENTARY UNITS

SELECTING AND SHARPENING A DRILL

UNIT 5

```
%
N1691G00G75X0.3536Y0.3536T10MG
N2X0.0Y0.0Z0.0
N3Z0.05
N4G01Z-0.05F10.0
N5G01X0.7071Y0.7071F20.
N6G03Y-2.1214I-1.0607J-1.
N7G01X-0.7071Y0.7071
N8G02Y0.7072I-0.3536J0.3
N9G00
N10G00X1.6464Y1.1464Z0.
N11Z0.05
N12G01Z-0.05F10.0
N13G03X-0.5Y0.5I-0.5J0.0F
N14G01X-3.0
N15G03X-0.5Y-0.5I0.0J-0.5
N20G01Y3.0
N21G00
N22G90G00X-2.0y-1.5M2
```



INTRODUCTION

Selecting the proper drill can be the difference between the right or wrong sized hole. Job specifications and the material you are drilling will determine which drill to choose. This unit will help you learn to choose and sharpen a drill.

UNIT OBJECTIVE

After completing this unit, you should be able to select and sharpen the correct drill for a job.

SPECIFIC OBJECTIVES

After completing this unit, you should be able to:

1. Identify various types of drills and their uses.
2. Select the correct drill to be used for an operation.
3. Perform a drilling operation using the correct drill.
4. Sharpen a twist drill using a pedestal grinder.

Put a check beside the statement that is true for you.

☐ **I can do these tasks and want to go through the Performance Checklist.**

SEE YOUR INSTRUCTOR

☐ **I want to study the information in this unit before I do the Performance Checklist.**

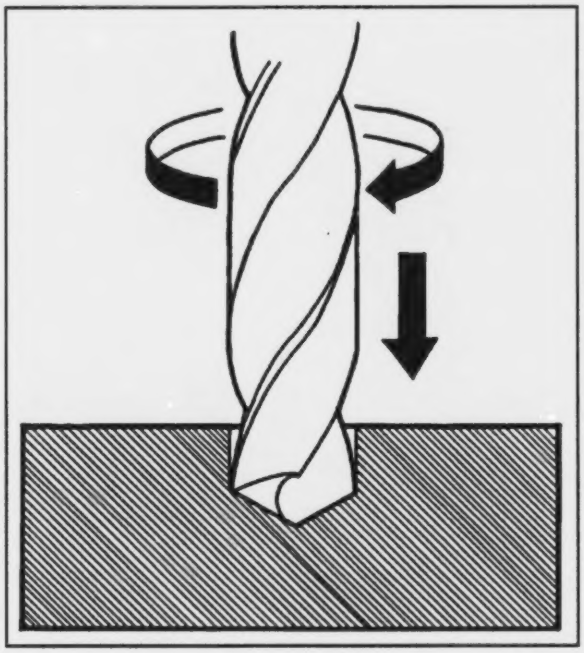
TURN PAGE AND BEGIN

A. TERMS

1. **Cutting edges -** the cutting part of a drill.
2. **Twist drill** - a cutting tool used to make holes in a workpiece.
3. **Shank** - the portion of the drill that mounts into the chuck or spindle. It may be straight or tapered.
4. **Body** - the portion of the drill bit between the point and the shank.
5. **Chuck** - moveable jaw mechanism for holding twist drills with straight shanks.
6. **Chuck key** - a device used to tighten the chuck after the drill is determined to be centered and running true.
7. **Flutes -** spiral grooves in the body of a drill that remove chips from the hole and permit cutting fluid to reach the cutting portions of the drill.
8. **Drill point gage** - a tool used to measure the angle of a drill point.
9. **Drill angle** - the angle of the drill cutting edges. It is usually expressed as the included angle.

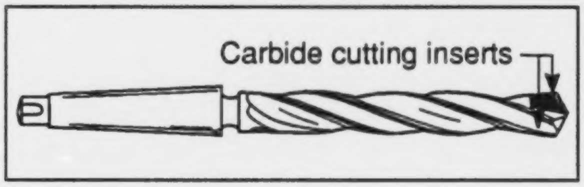
B. TWIST DRILL MATERIALS

Most drill bits are made from one of three materials. Each material is best suited to certain drilling jobs.



1. **Carbon steel** drills are fine for slow-speed jobs. Overheating, however will cause the drill to dull. These are seldom used in metal working.

2. **High-speed steel** drills are the most common type. They keep a sharp cutting edge at high speeds through many metals.

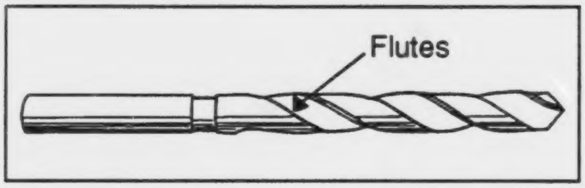


3. **Tungsten carbide** drills are

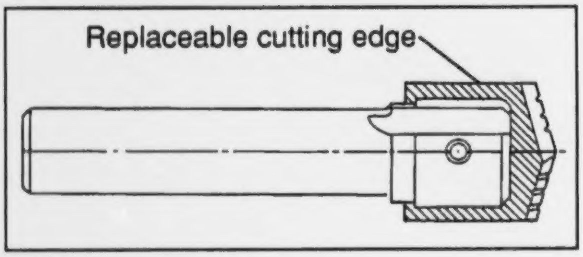
material, glass-bonded plastics, etc. They have carbide cutting tips bonded to the end of the drill.

C. TOOLS FOR MAKING HOLES

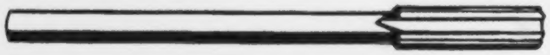
1. **Twist** drills are the type you will use most often in a machine shop. They have flutes (grooves) along the body to allow chips from the cutting action to be brought to the surface. The flutes also permit cutting fluid to reach the cutting portion of the drill.



2. **Spade** drills have flat, replaceable blades. Their holes, however, tend not to be as exact and true. They are commonly used to drill very large holes.



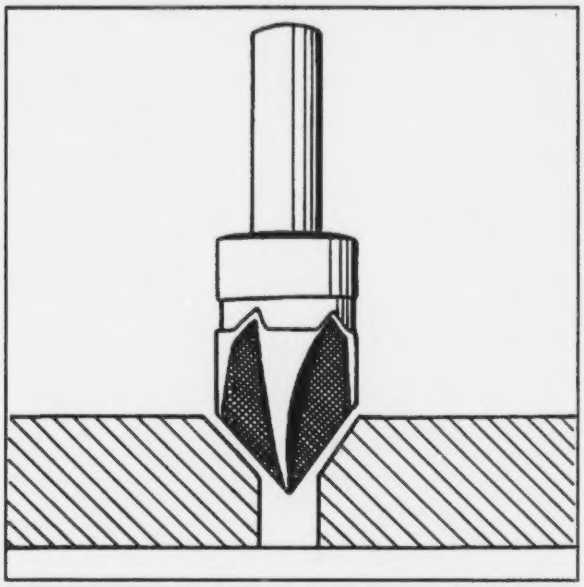
3. **Reamers** are used to finish a drilled hole to an exact diameter and give it a smooth finish.

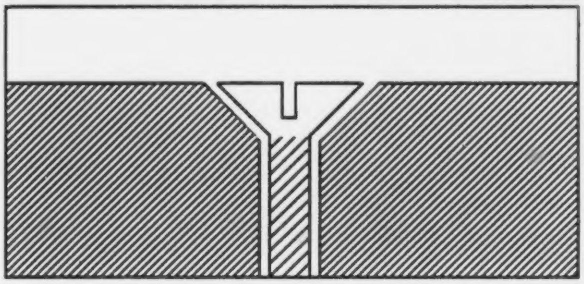


Straight-shank reamer

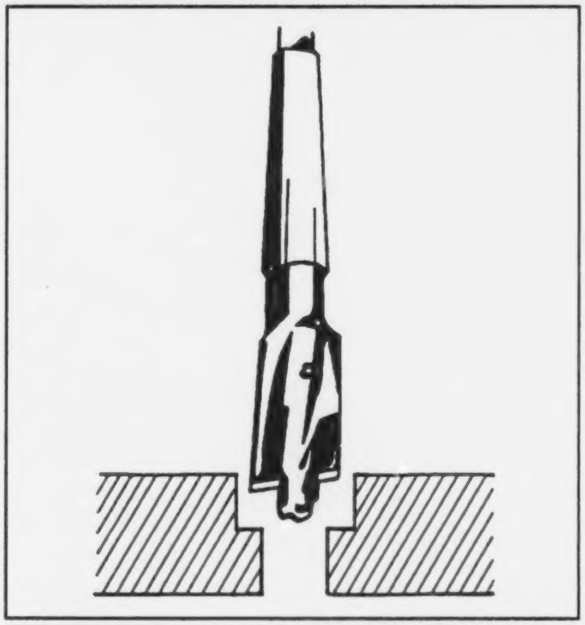
4. **Other hole-making tools.** Certain jobs require that a hole be more than just straight and round. Special tools can create the effects you want.

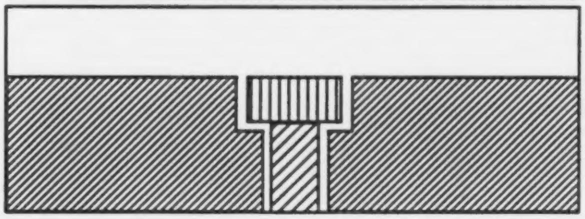
a. Countersink--a cone-shaped opening at the outer surface. This can be used to remove a burr from the edge of a drilled hole or to provide a seat for a tapered-head screw.





b. Counterbore--enlarges and recesses a hole so it will accept a socket-head capscrew.





DO SKILL SHEET 1

DO ACTIVITY SHEET 1

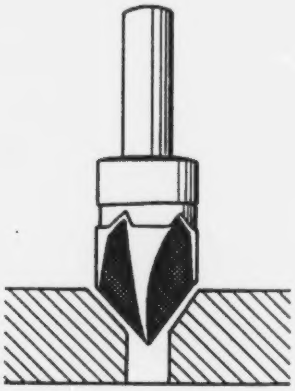
SELECTING AND SHARPENING A DRILL

SKILL SHEET 1

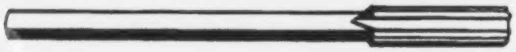
Match the pictures of the cutting tools below with their correct names.

A.	twist drill	D.	counterbore
B.	spade drill	E.	countersink
C.	reamer		

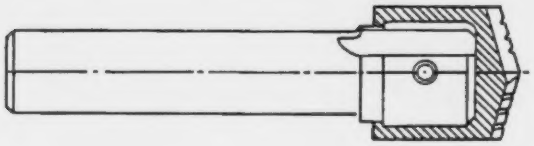
___ 1.



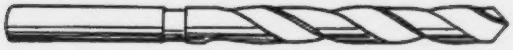
___ 2.



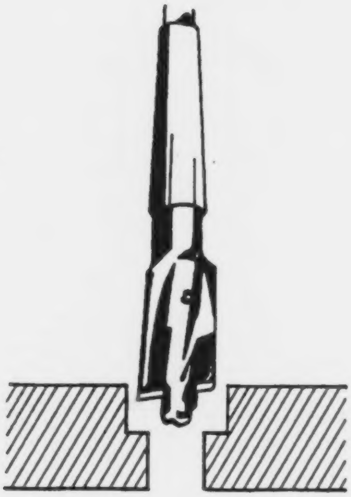
___ 3.



___ 4.



___ 5.



SELECTING AND SHARPENING A DRILL

ACTIVITY SHEET 1

Your instructor will provide you with several types of cutting tools. See if you can identify them.

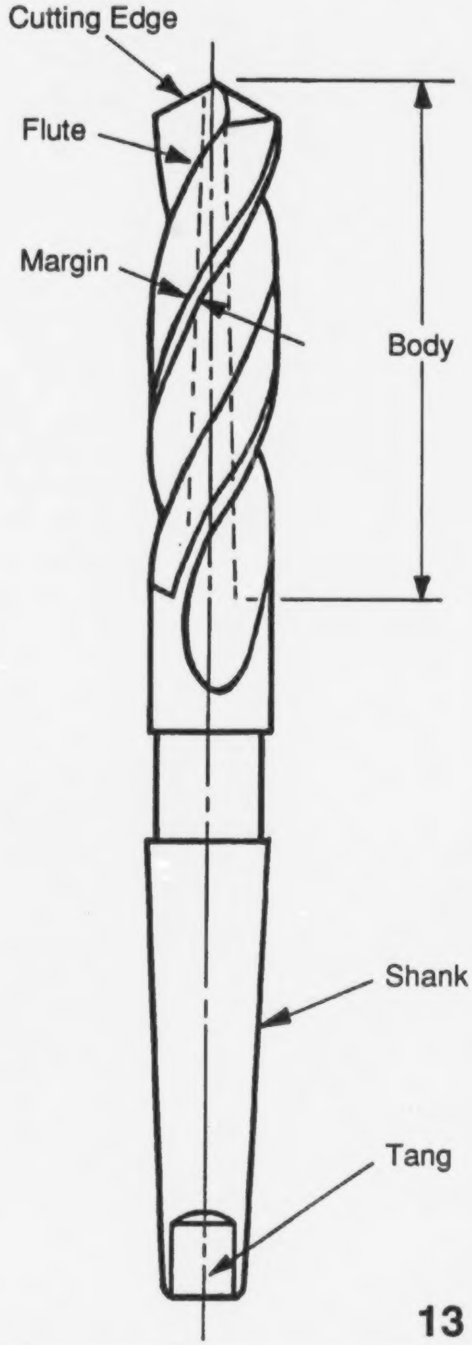
D. PARTS OF A TWIST DRILL

1. The **cutting edges** of the drill actually cut the workpiece material.

2. Twist drills have **shanks** that are either straight or tapered. The shank fits into the slot in the drill press spindle and holds the bit in place. The shank may be held in place by a drill chuck.

3. The **body** is the portion between the point and the shank.

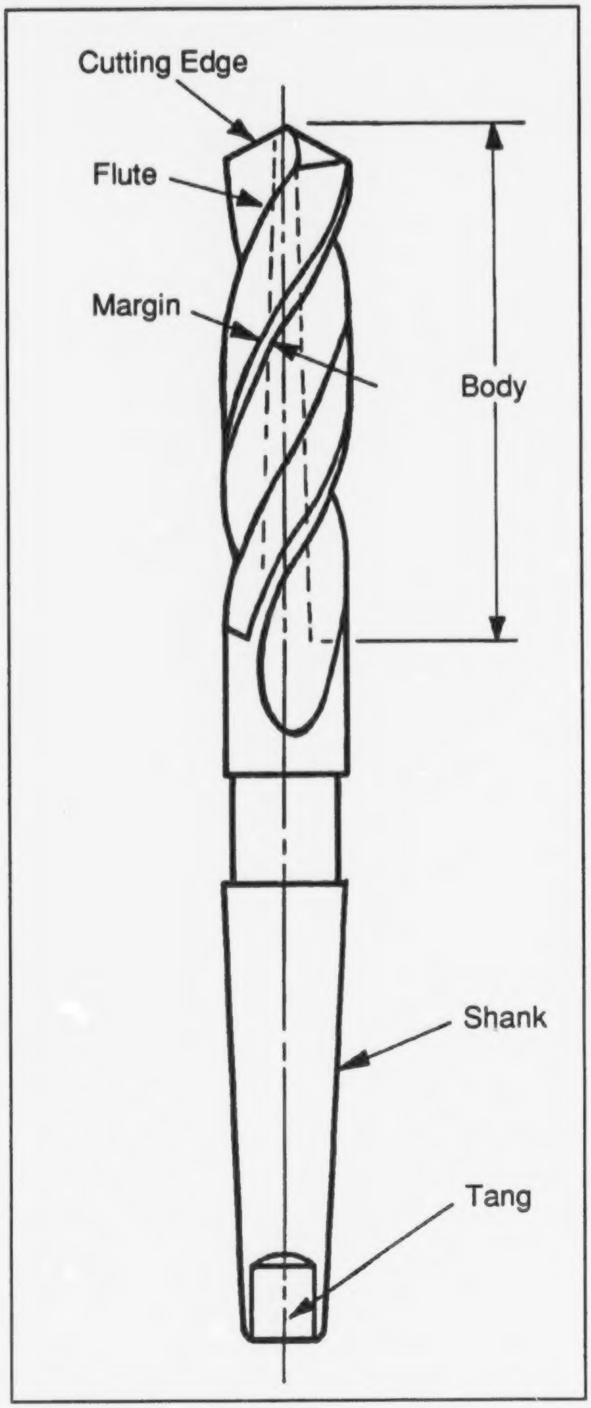
4. The **tang** on a taper shank fits into a slot in the spindle,



sleeve or socket and keeps the tool from slipping in the socket.

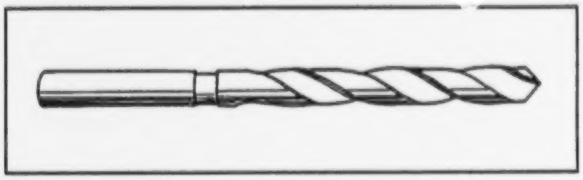
5. Two or more spiral groves run the length of the drill body. They are called **flutes.** The flutes are channels through which the chips can escape from the hole being drilled. They also allow lubricant and coolant to get down to the cutting edge.

6. The **margin** is a narrow strip extending back the entire length of the flute. It is the full diameter of the drill.

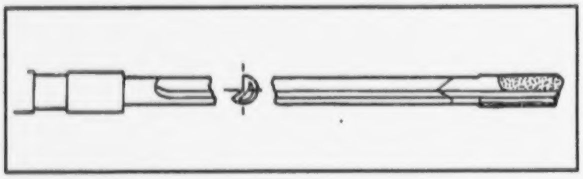


E. TYPES OF TWIST DRILLS

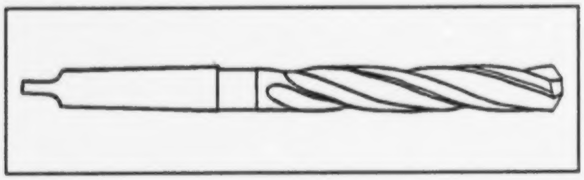
1. The **two-flute** twist drill is the most commonly used bit for all-purpose drilling.



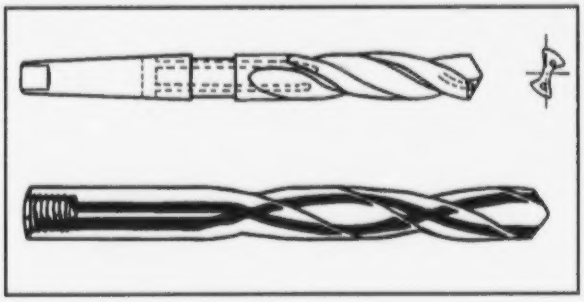
2. The **straight-flute** drill produces short chips. It is best for brass, other nonferrous metals, and brittle plastics.



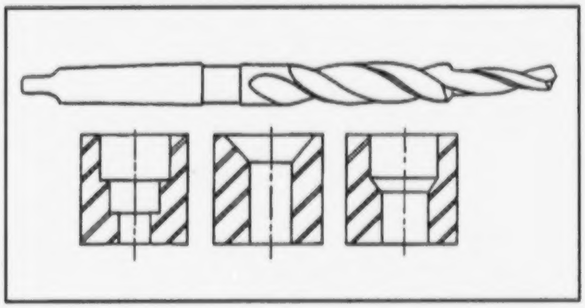
3. **Three-** and **four-flute drills** are used when holes need to be made larger or when exact size is important. They drill more accurately than other types.



4. The **oil-hole** drills have coolant holes through their bodies to permit fluid or air to reach the point.



5. The **step** drill allows a two-step drilling process (two sizes of holes) to be done in one operation.



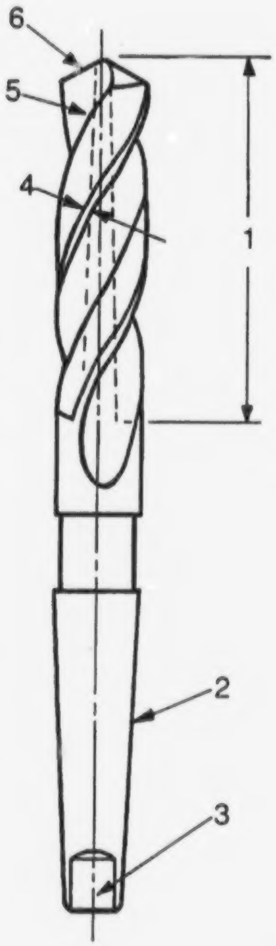
DO SKILL SHEET 2

SELECTING AND SHARPENING A DRILL

SKILL SHEET 2

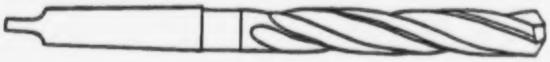
Part 1. Identify the parts of the twist drill below using these words:

lip
tang
flutes
margin
shank
body

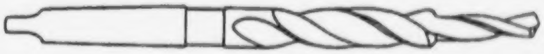


Part 2. Identify these twist drills by writing their names beneath the pictures.

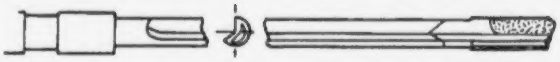
8.



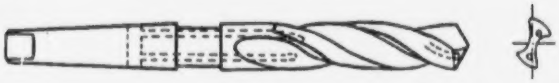
9.



10.



11.



F. DRILL SIZES

Drills are sized in three ways in the inch-standard system.

1. **Wire number**. The wire numbers of drills go from #80 (.0135") to #1 (.040").
2. **Letter sizes** are given from A (.234") to Z (.413"). They are larger than the number-size drills.
3. **Fractional size** begins at 1/64" diameter. Each successive drill is 1/64" larger.

Number (wire)	Drill Size Letter	Designation Fractional (inch)	Metric (mm)	Decimal Equivalent
1				0.2280
			5.8	0.2283
			5.9	0.2323
	A			0.2340
		15/64		0.2344
			6.0	0.2362
	B			0.2380
			6.1	0.2402
	C			0.2420

Drills are sized by millimeters in the metric standard system.

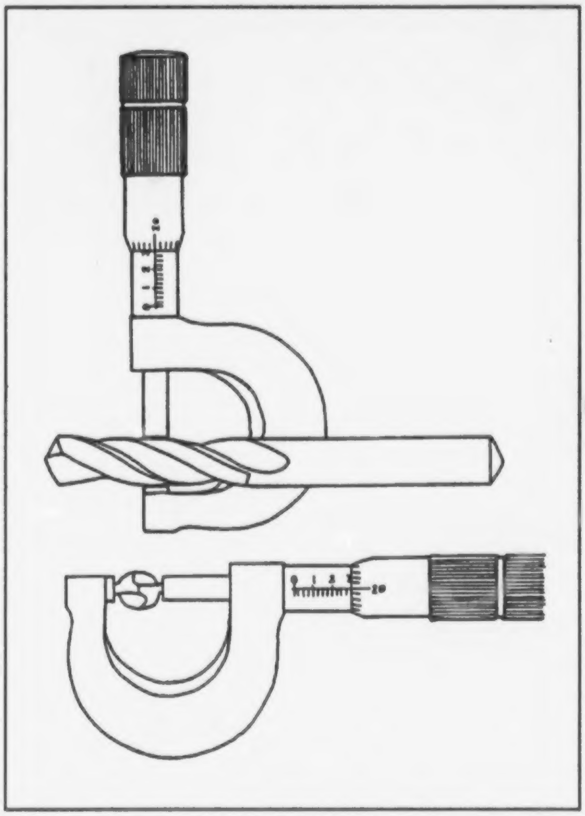
G. DETERMINING DRILL SIZE

ALWAYS CHECK THE DRILL DIAMETER BEFORE YOU USE IT.

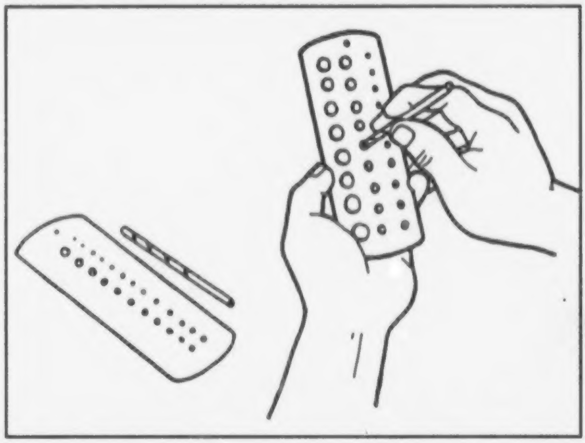
Number (wire)	Drill Size Letter	Designation Fractional (inch)	Metric (mm)	Decimal Equivalent
1				0.2280
			5.8	0.2283
			5.9	0.2323
	A			0.2340
		15/64		0.2344
			6.0	0.2362
	B			0.2380
			6.1	0.2402
	C			0.2420

Manufacturers' charts supply the drill sizes. The drills also usually have the diameters stamped on the shank.

You can use a micrometer to measure the diameter of the drill. Be sure to measure across the margin of the drill in a place where no cutting wear is visible.



You can also check the diameter by using a **drill gage.**



DO SKILL SHEET 3

DO ACTIVITY SHEETS 2& 3

SELECTING AND SHARPENING A DRILL

SKILL SHEET 3

Using a manufacturer's drill chart, give the correct letter, fractional or number designation for each drill size listed below:

	Size	Designation (answer)
1.	.0135	1.
2.	.0156	2.
3.	.257	3.
4.	.5937	4.
5.	.2810	5.
6.	.0292	6.
7.	.3320	7.
8.	.0595	8.
9.	.2344	9.
10.	.625	10.

SELECTING AND SHARPENING A DRILL

ACTIVITY SHEET 2

Your instructor will provide you with a micrometer and 10 drills with the shank marking covered over with tape. Measure each drill and write its decimal measurement. Then convert that to fractional, number, letter, or metric to give its true designation.

	MEASURED SIZE	DESIGNATION
1.		
2.		
3.		
4.		
5.		
6.		
7.		
8.		
9.		
10.		

SELECTING AND SHARPENING A DRILL

ACTIVITY SHEET 3

Equipment:

- Part drawing (provided)
- Drill press with vise
- Complete drill index with assortment of drills (number, fractional, letter)
- Combination square
- Tap drill charts
- Aluminum or steel workpiece
- #3 centerdrill
- Layout dye
- Scriber
- Center punch
- Table of cutting speeds for drilling

Procedure:

Evaluation:

3 2 1 N

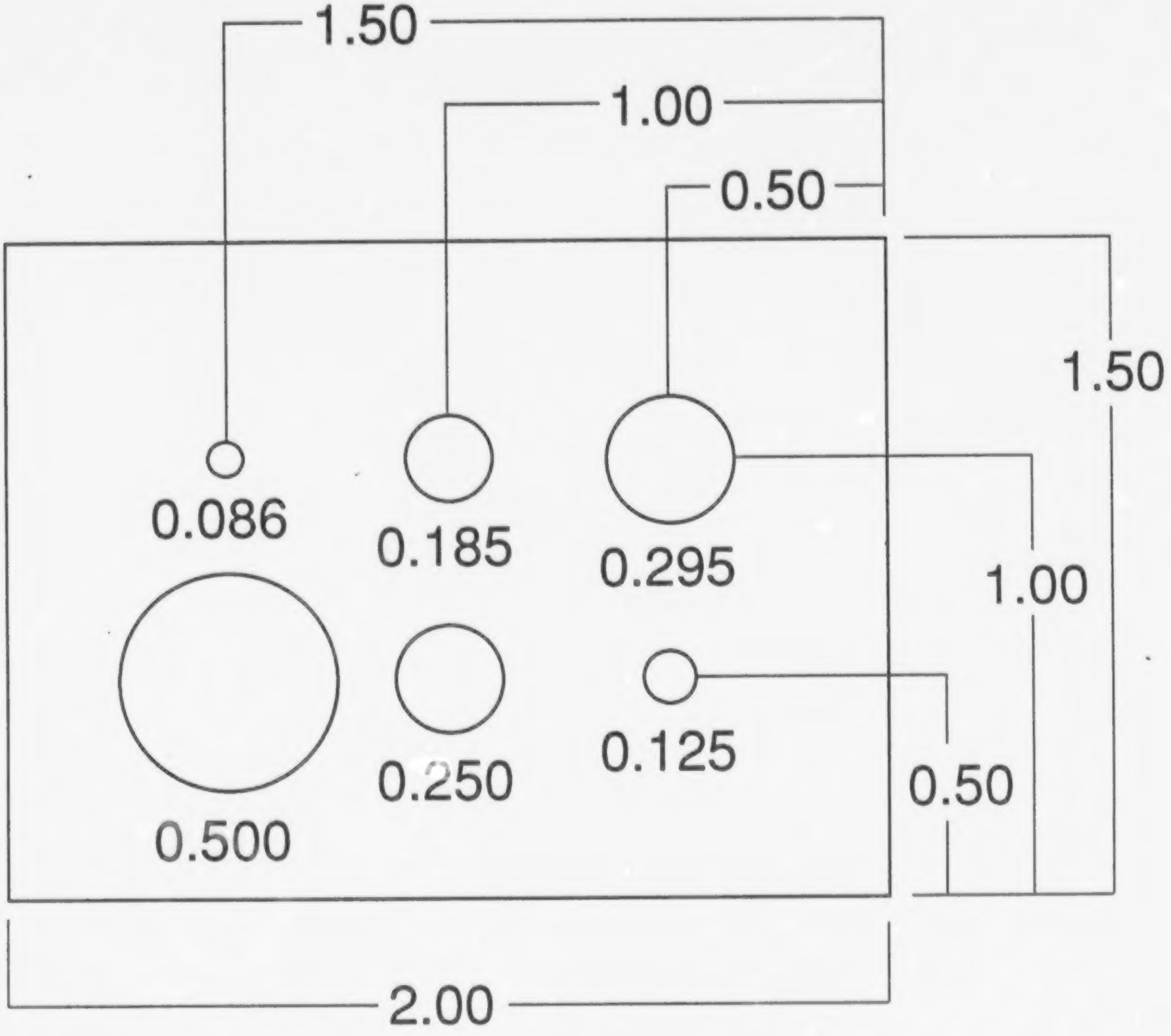
1. Select drills to create the hole sizes indicated on the drawing. __ __ __ __
2. Layout part hole center. Center punch each hole. __ __ __ __
3. Calculate proper spindle rpm for each drill diameter. __ __ __ __

4. Counterdrill each punched location. ____

5. Place part in vise. Drill each hole through part as shown on drawing. ____

6. Deburr each hole, top and bottom. ____

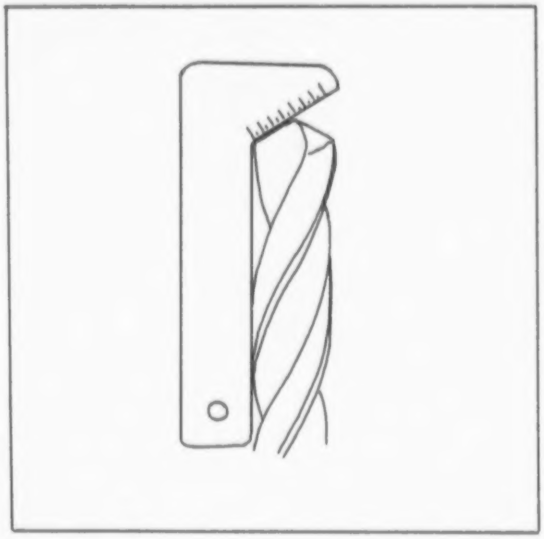
7. Present part to instructor for inspection. ____


1.50
1.00
0.50
1.50
1.00
0.50
0.086
0.185
0.295
0.500
0.250
0.125
2.00

171

H. SHARPENING A TWIST DRILL

You use a pedestal or bench grinder to sharpen a drill. Be sure to review the operation of the pedestal grinder. Don't use the grinder without your instructor's permission.



When you sharpen a twist drill, you do two things:

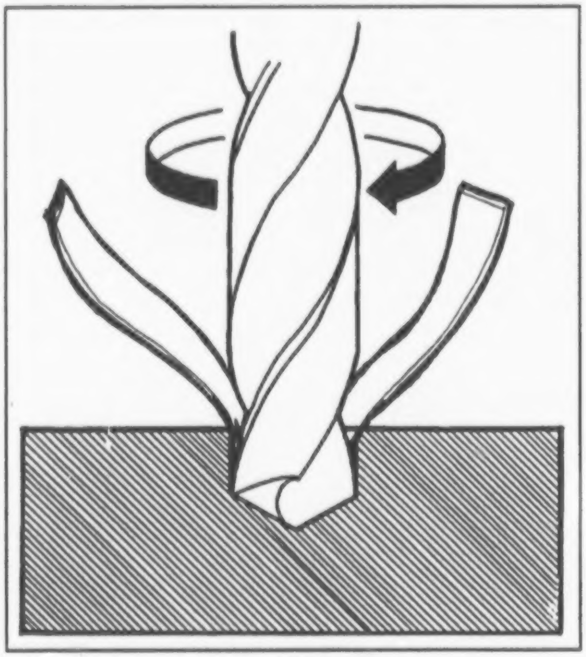
1. You make the cutting edges sharp.
2. You give the cutting edges the proper angle.

Commonly a twist drill is sharpened to an included angle of 118^{0}.

This angle is measured with a tool called a drill point gage.

For the drill to cut properly, the cutting edges must be the same length.

You can test the cutting edges by drilling into a piece of soft material such as aluminum. If the two chips produced are the same size and the drill cuts easily, the drill has been properly sharpened.



The sharpened cutting edge or lip on the end of the drill must be the highest point. You do this by grinding away material behind the cutting edge.

DO ACTIVITY SHEET 4

SELECTING AND SHARPENING A DRILL

ACTIVITY SHEET 4

Equipment:

Twist drill 1/2" or larger	Drill point gage
Pedestal grinder and depressor	Water pot
Face shield	

CAUTION: WEAR FACE SHIELD THROUGHOUT THIS ACTIVITY

Procedure:	Evaluation: 3 2 1 N
1. Have your instructor assist you to dress the grinding wheel.	_ _ _ _
2. Adjust tool rest on the grinder to within 1/16" of wheel face. Be sure grinder is off and wheel is not turning.	_ _ _ _
3. Turn on the grinder. Grasp drill at the shank.	_ _ _ _
4. Sharpen one cutting lip at a time. Place lip against wheel face at 59°. (This is one half of the 118° included angle.) Slowly rotate the drill away from the lip. Quench in water to keep cool as needed. Be careful not to overheat	

the drill or burn yourself. _ _ _ _

5. Check angle with drill gage. _ _ _ _

6. Check to see that lips are the same length by drilling a hole in a piece of soft material such as aluminum. _ _ _ _

7. Have your instructor check your work. _ _ _ _

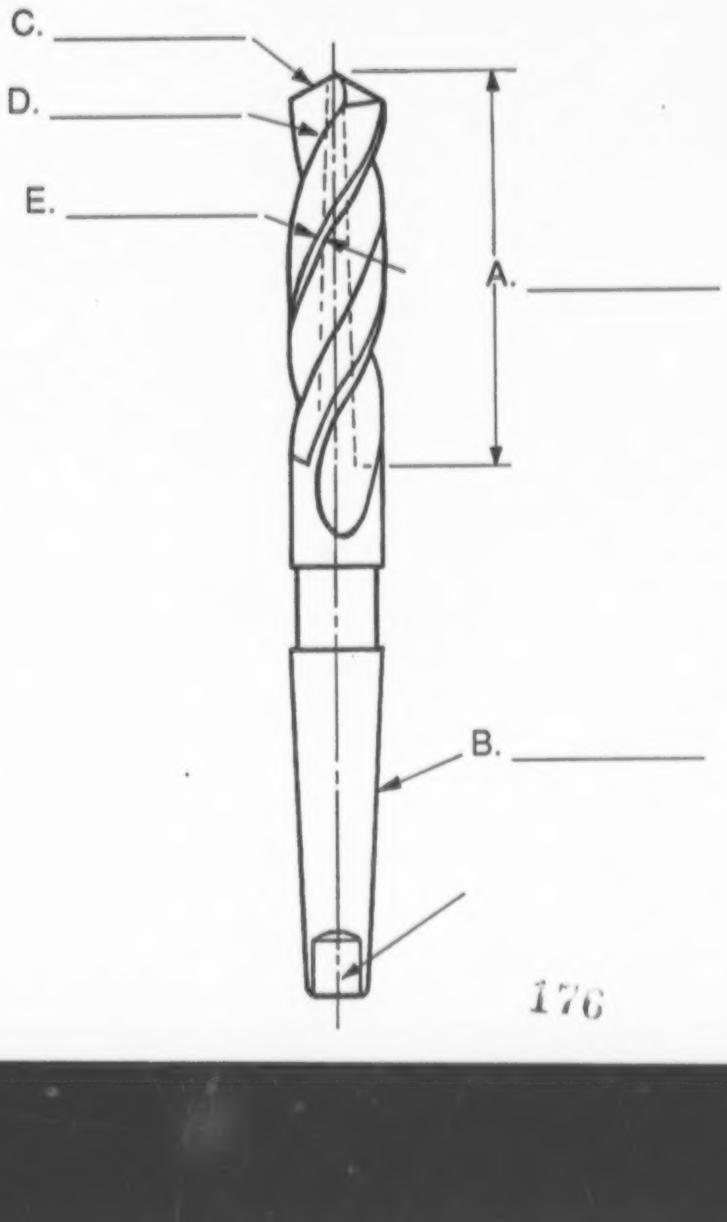
8. Return tools and materials to their proper places. _ _ _ _

SELECTING AND SHARPENING A DRILL

UNIT TEST

Part 1. Identify the parts of the twist drill bit below using these words:

lip
tang
flutes
margin
shank
body



Part 2. Match the terms below with their definitions:

A.	Flutes	D.	Shank
B.	Body	E.	Cutting edges
C.	Twist drill		

___ 7. The cutting part of a drill.

___ 8. A cutting tool used to make holes in a workpiece.

___ 9. The portion of the bit that mounts into the drill chuck or spindle. It may be straight or tapered.

___10. The portion of the drill bit between the point and the shank.

___11. Spiral grooves in the body of a drill that remove chips from the hole and permit cutting fluid to reach the cutting portions of the drill.

Part 3. Choose the best response for each question.

___12. Which three materials below are used for drills?

A.	Carbon steel	D.	Carbide
B.	Aluminum	E.	High-grade plastic
C.	Brass	F.	High-speed steel

___13. What tool would be used to produce a seat for a socket-head capscrew?

A.	Countersink	C.	Counterbore
B.	Tap	D.	Twist drill

___14. What tool would be used to deburr a drilled hole?

A.	Counterbore	C.	Twist drill
B.	Countersink	D.	Gun drill

___15. On the drill below, where would you measure with a micrometer to determine drill size?

SELECTING AND SHARPENING A DRILL

PERFORMANCE CHECKLIST

Student____________________________has successfully performed the following tasks:

	YES	NO	COMMENTS
1. Identified various types of drill bits and their uses.			
2. Selected the correct drill bit to use for a particular operation.			
3. Performed a drilling operation using the correct drill bit.			
4. Sharpened a twist drill using a pedestal grinder.			

Satisfactory - Should move on ☐
Repeat this unit ☐

____
Student's Signature/Date

____
Instructor's Signature/Date

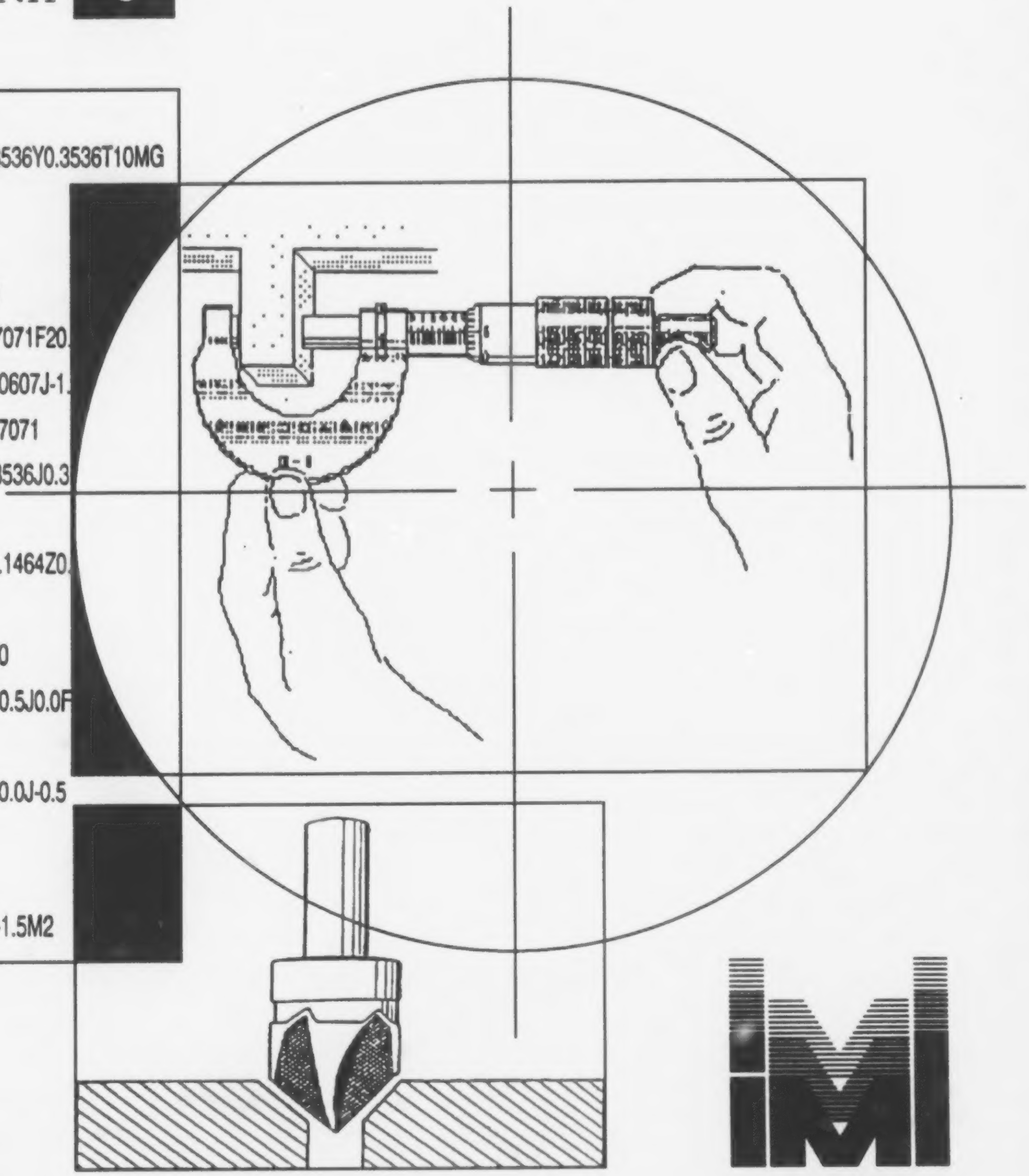
Machine Shop

90-9200-S
SUPPLEMENTARY UNITS

USING A CNC MACHINE

UNIT 6

```
%
N1691G00G75X0.3536Y0.3536T10MG
N2X0.0Y0.0Z0.0
N3Z0.05
N4G01Z-0.05F10.0
N5G01X0.7071Y0.7071F20.
N6G03Y-2.1214I-1.0607J-1.
N7G01X-0.7071Y0.7071
N8G02Y0.7072I-0.3536J0.3
N9G00
N10G00X1.6464Y1.1464Z0.
N11Z0.05
N12G01Z-0.05F10.0
N13G03X-0.5Y0.5I-0.5J0.0F
N14G01X-3.0
N15G03X-0.5Y-0.5I0.0J-0.5
N20G01Y3.0
N21G00
N22G90G00X-2.0y-1.5M2
```



USING A CNC MACHINE

INTRODUCTION

When you perform a conventional machining operation, you study the drawing, mount and position the workpiece on the machine and then use your hands to guide the workpiece through the operation. With a computer numerical control (CNC) machine, all of the machine movements are controlled by instructions from a punched tape or a computer. It is important for today's machinist to understand how to program and use a CNC machine.

UNIT OBJECTIVE

After completing this unit, you should be able to operate a CNC milling machine tool and make a simple part.

SPECIFIC OBJECTIVES

After completing this unit, you should be able to:

1. Define terms associated with CNC machine operation.
2. Identify X, Y & Z axes.
3. Using a given part, list the operations that must be performed to complete the part.

4. Enter data for a part into the CNC controller.
5. Test the program.
6. Set up the cutting tool and workpiece.
7. Machine and inspect the part.

Put a check beside the statement that is true for you.

☐ **I can do these tasks and want to go through the Performance Checklist.**

SEE YOUR INSTRUCTOR

☐ **I want to study the information in this unit before I do the Performance Checklist.**

TURN PAGE AND BEGIN

A. TERMS

1. **Compter numerical control** - a system in which a computer controls a machine by using commands in a numerical (number) format.
2. **Conventional machining** - machining by manual control; a machinist directly controls movement of the machine.
3. **Machine controller**--a computer that controls the movement of a machine tool.
4. **Servo-motor** - usually a direct-current motor that responds to pulses from a machine controller to cause motion along some axis of the machine tool.
5. **Part program** - the set of coded instructions converted by the machine controller into machine motion to produce a part.
6. **Programmer** - an individual who writes coded instructions to control CNC machine tools.
7. **Repeatability** - ability of a machining system to return to the same coordinate location time after time with very little error.
8. **Axis** - a direction along which movement of a tool or workpiece may take place (X, Y, Z).
9. **Axis signs** - plus or minus signs to indicate the direction of motion along a machine axis.

10. **Coordinate** - a specified point, indicating a location in X, Y and Z space.

11. **Origin** - the coordinate location with the value X0, Y0 and Z0).

12. **Process planning** - the activity in which the part programmer decides the order of operations, the tools to be used, etc. for making a certain part.

13. **Incremental dimensioning** - a dimensioning or control system in which each new dimension is taken from the last dimension or machine movement.

14. **Absolute dimensioning** - a dimensioning or control system in which each new dimension is taken with respect to a common origin point.

15. **Home position** - a coordinate location usually designated by the operator used when changing a tool or workpiece.

16. **Machine zero** - hard reference point that is used by the machine controller to locate itself upon power-up.

17. **Program zero** - the reference coordinate from which the part program coordinates are taken.

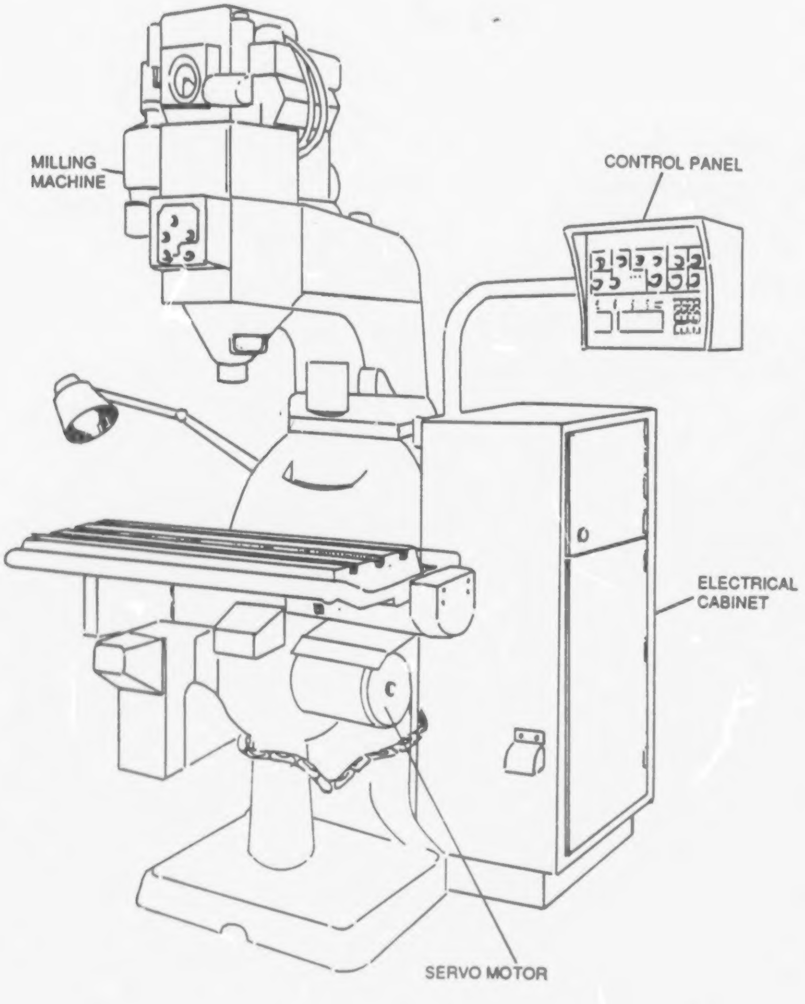
18. **Z-clear** - vertical distance above workpiece that will allow the tool to clear while moving from one position to the next.

19. **MDI** - manual data input; mode for entering data to the controller manually.

20. **Manuscript** - hand-written CNC program on paper.
21. **JOG**--mode that allows operator to position machine axis by pressing axis jog switches.

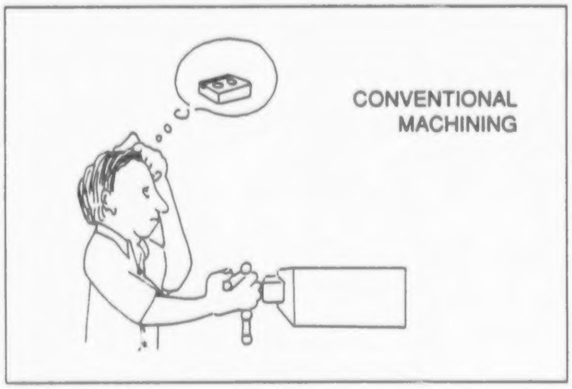
B. EQUIPMENT

PARTS OF A CNC MILLING MACHINE


MILLING MACHINE
CONTROL PANEL
ELECTRICAL CABINET
SERVO MOTOR

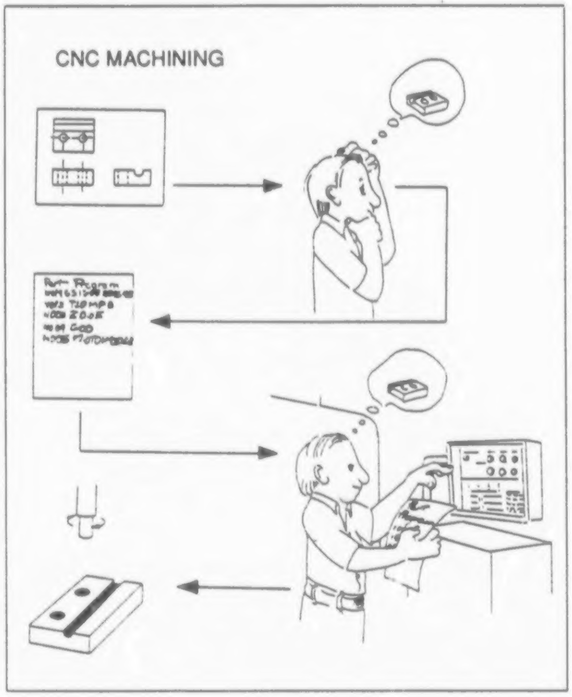
C. PRINCIPLES OF CNC OPERATION

In conventional (regular) machining, a highly skilled machinist controls the movement of the tool and the workpiece.



He or she examines the part drawing then carefully adjusts the machine dials, the tool and the workpiece location to remove just the right amount of material in just the right places. The machinist controls the machine to produce the required part.

With computer numerical control (CNC), the job of the machinist in the process changes. A computer, called a machine controller, replaces the machinist in guiding machine motions.



BEST COPY AVAILABLE

The machinist becomes the programmer. He or she uses numbers to tell a computer how to guide the machine.

Machinists/programmers must be good at reading a print and understanding what it means. They must understand the machine, cutting tools, feeds and speeds. Programmers must be able to "see" the whole process so they can tell the computer what to tell the machine to do.

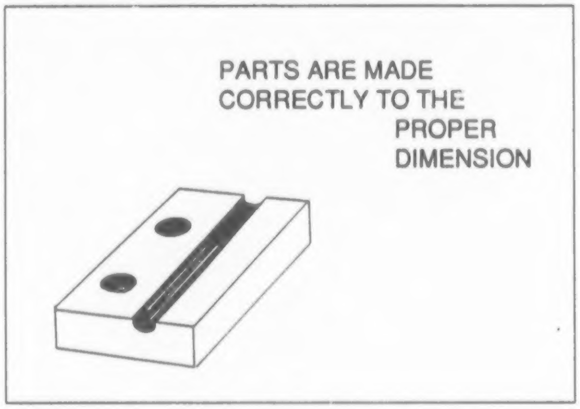
D. ADVANTAGES OF CNC MACHINING

Automated machining with a CNC system gives three advantages:

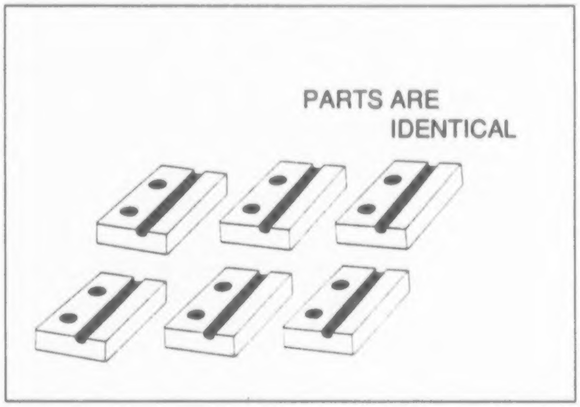
1. **Speed**. CNC machines can do the work faster than an operator. Operators may hesitate or take extra time in

making adjustments. Under computer control, the machine moves without hesitation.

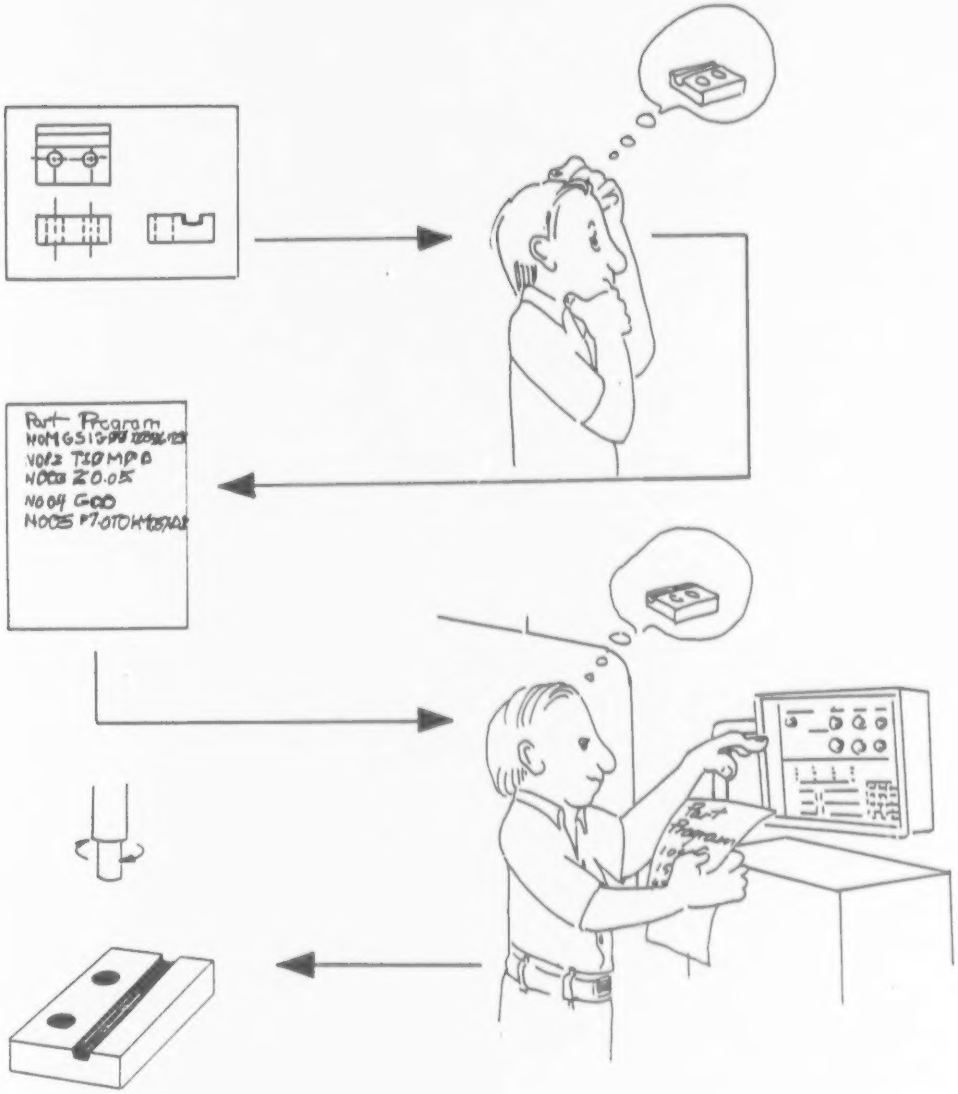
2. **Accuracy**. A properly programmed CNC machine can produce parts that are more accurate (free from error) than can a human operator.


PARTS ARE MADE CORRECTLY TO THE PROPER DIMENSION

3. **Repeatability**. A well-maintained machine, using the same program, will produce parts that are exactly alike over and over again. The computer doesn't get tired, or bored or become careless the way an operator may.


PARTS ARE IDENTICAL

E. DIAGRAM OF CNC OPERATING PRINCIPLE

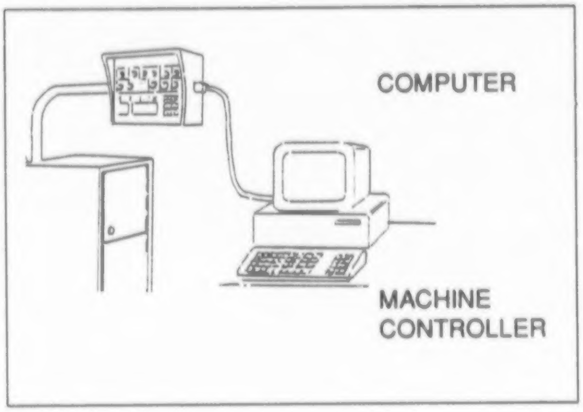

Part Program
N001 G513
N002 T10 M00
N003 Z0.05
N004 G00
N005
Part Program

F. A CNC PART PROGRAM

In a CNC machining process, a skilled programmer reviews the part drawing and writes a set of coded instructions called a part program.

The program may be written on paper or typed on a microcomputer. After the program is written, it is stored on a computer disk or punched onto a paper tape.

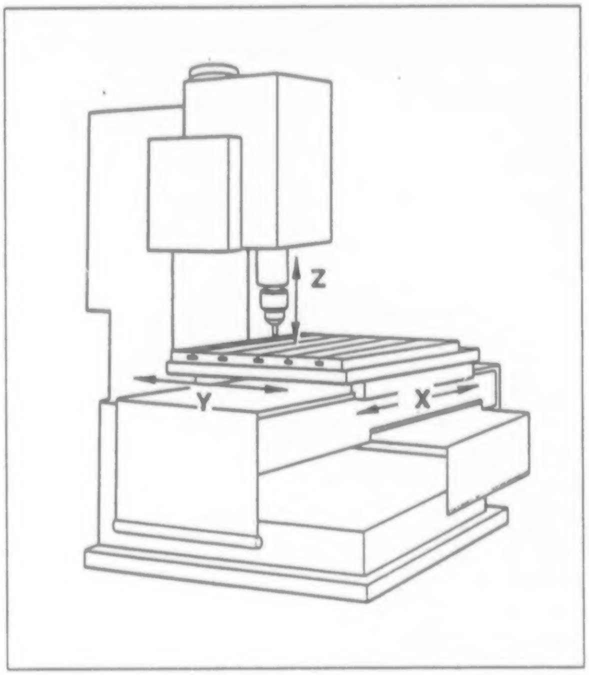
The program may also be loaded into the machine by hand. This is called "manual data input." That means the operator pushes keys on the control unit that are a series of letters, numbers and other characters that identify certain machine functions.



The program may be punched on a tape, which is then loaded into the machine control.

The program may be typed on a microcomputer and loaded into the machine directly from there.

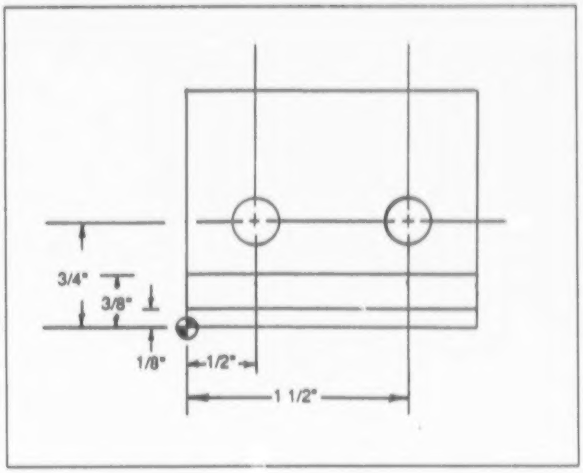
G. COORDINATE SYSTEM



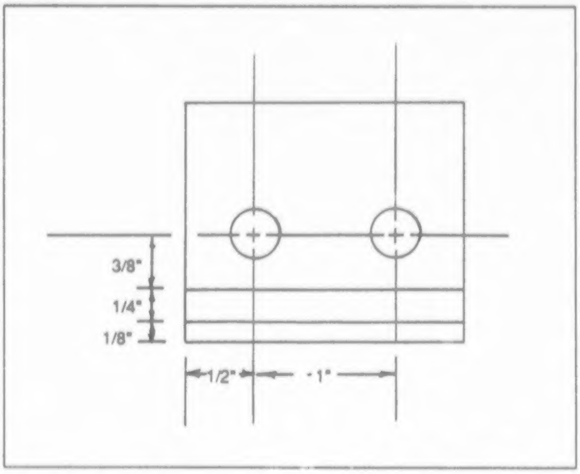
The part program instructs the machine where to position the workpiece and the tool. There has to be some way to define this movement. The system of directions that directs the CNC machine movements and locations through the part program is called a coordinate system.

There are two ways of dimensioning and moving the tool in CNC machining.

1. Absolute. In absolute dimensioning all coordinate locations are given from the point of origin. You look back to the origin point after each move and then determine where you will go next.



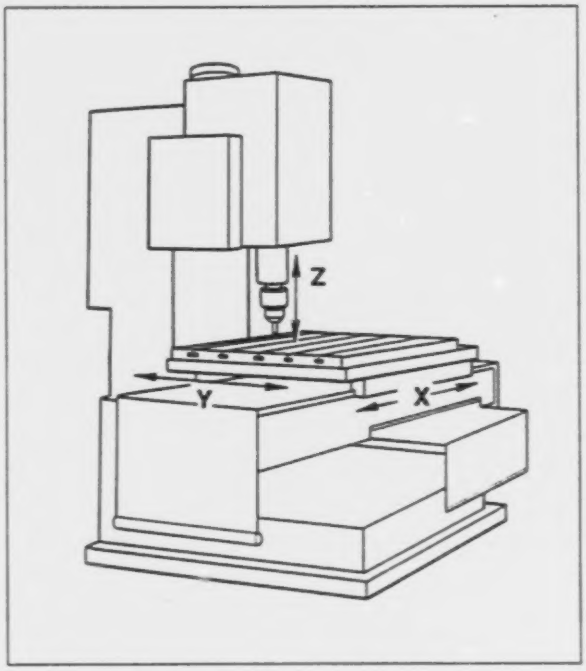
2. Incremental. In incremental dimensioning, the machine "forgets" where it has been after each movement. Each new location is known only in reference to the previous location.



H. THREE MACHINE AXES

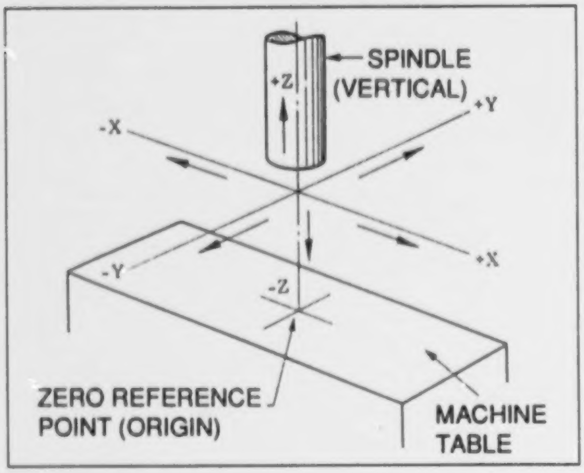
The coordinate system is based on three machine axes (X, Y and Z).

1. **X axis.** On a vertical machine (one where the spindle moves up and down), the X axis is horizontal (left and right) travel of the table.



2. **Y axis.** The Y axis is the cross movement of the saddle, in and out from the column. The X and Y axis cross each other.

3. **Z axis.** The Z axis is the motion of the spindle up and down.


SPINDLE (VERTICAL)
+Z
-X
+Y
-Y
+X
-Z
ZERO REFERENCE POINT (ORIGIN)
MACHINE TABLE

The direction of movement in any of those three axes is given by a plus or minus sign (- or +). The sign tells which direction to go.

The point where the axes come together is called the origin or zero point. The point of origin is indicated by X0, Y0 or Z0. Four sections (called quadrants) are formed when the X and Y axes cross. The plus (+) and Minus (-) sings indicate the direction from the 0 point.

DO SKILL SHEET 1

USING A CNC MACHINE

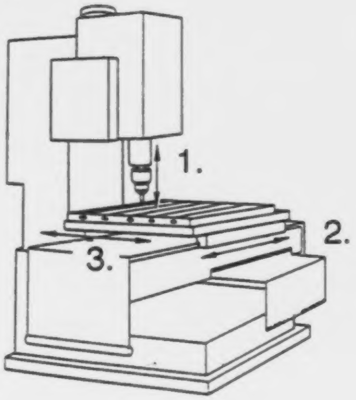
SKILL SHEET 1

Label machine axes indicated by the arrows on the drawing below.

1.

2.

3.

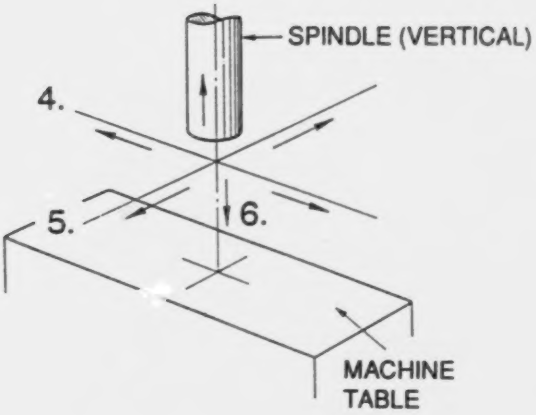


Label the axes on the drawing

4.

5.

6.


SPINDLE (VERTICAL)
4.
5.
6.
MACHINE TABLE

Match the letters to the parts of this CNC machine.

A. Electrical panel

B. Control panel

C. Servo motor

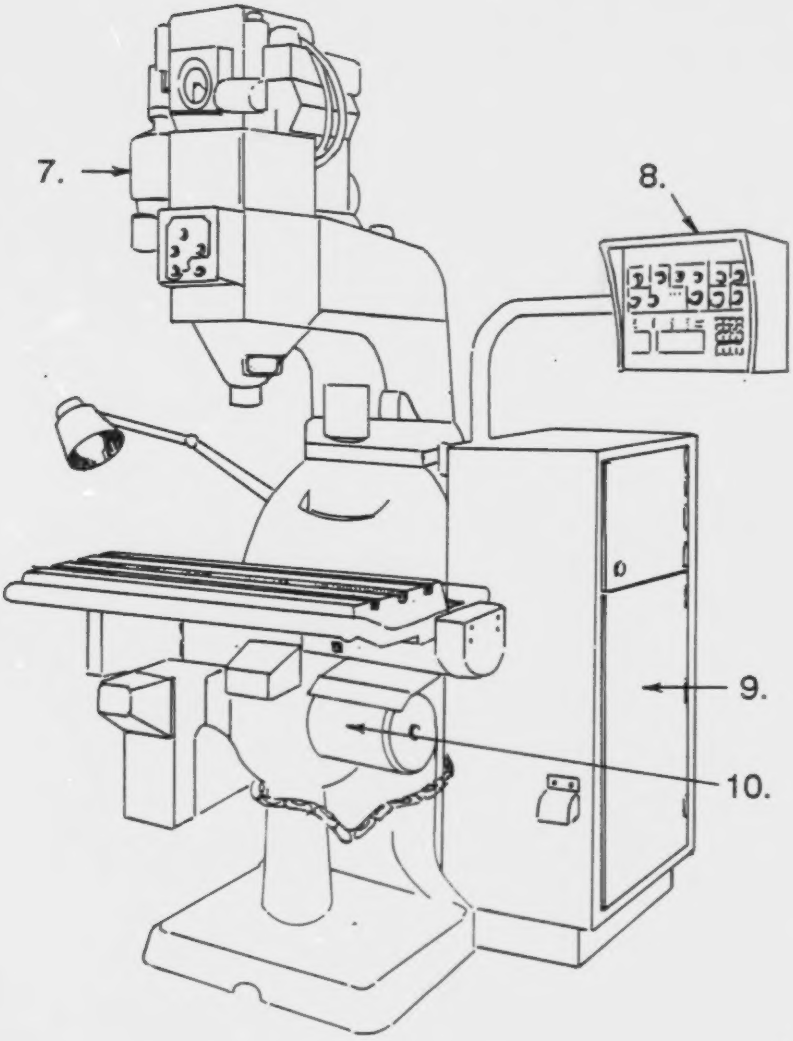
D. Milling machine

___ 7.

___ 8.

___ 9.

___ 10.



Answer the questions below.

11. List two advantages of a CNC machine.

A.

B.

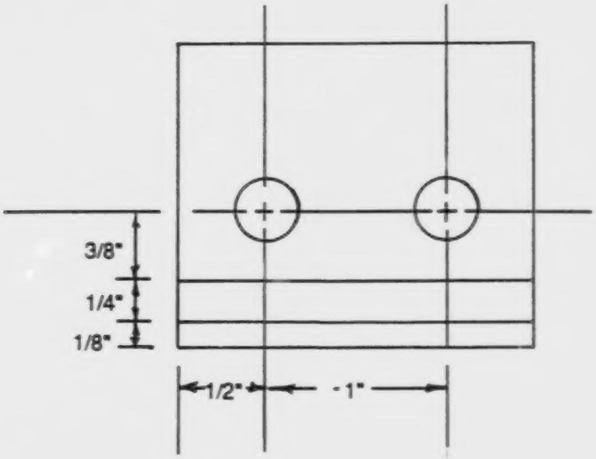
12. The point where three machine axis intersect (come together) is called the__________________________.

13. Axis direction is designated with two signs. What are they?

A.

B.

14. Is the part below dimensioned incremental or absolute?



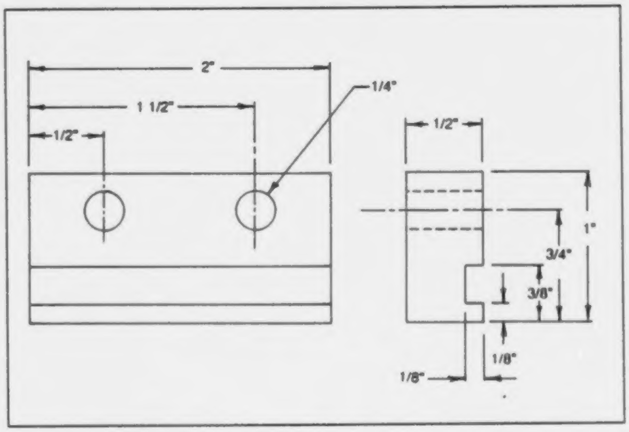
15. List two ways of storing a part program.

A.

B.

I. PLANNING THE CNC PROCESS

Planning for CNC machining involves several steps.



1. Study the part drawing thoroughly. Be sure you understand what it is calling for. Be sure you know what operations will be performed.

2. Determine the types of tools, toolholders and work-holding devices you will need.

OPERATIONS:

1) MILL
2) DRILL

3. Visualize the actual cutting motions and table movements that will need to take place on the machine.

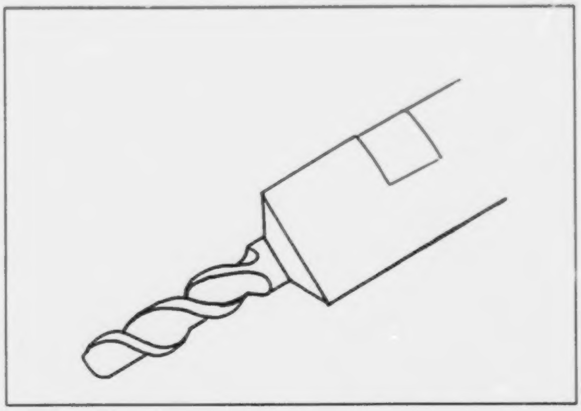
TOOLS:
TO1 DRILL
TO2 MILL

For example, a small hole would first require a center-drilling operation. This would be followed by drilling to finish the hole to size.

A slot would require an endmill of the same size as the slot

4. Ask yourself, "What do I do first; what do I do next?" etc.

5. Once you've decided on the right tools, you calculate proper spindle speed and feedrate for the particular tools you will be using. Refer to the unit on "Speeds and Feeds" for review if necessary.



To simplify matters, on the part you will produce the only tool necessary is a 1/4-inch, two-flute endmill. You will use the endmill to make the slot and drill the two holes. We will call this tool by its CNC coded name, "T01," or tool number one.

"T01"
TOOL # 1

Here are the formulas you will need to calculate spindle speed and feedrate:

$$\text{Spindle Speed (S)} = \frac{\text{Cutting Speed X 12}}{\text{pi (3) X cutter diameter}} = \text{RPM}$$

Feedrate = R X T X RPM

R = feed per tooth (from table)

T = number of teeth

DO SKILL SHEET 2

USING A CNC MACHINE

SKILL SHEET 2

Using the part drawing given, list the operations that must be performed to complete the part. Assume that you have a workpiece that is already the finished size.

1. Calculate the feedrate and spindle speed RPM for the 1/4-inch, 2-flute endmill. Give the feedrate in inches-per-minute. Assume you will make the part from aluminum (cutting speed = 400 rpm and .010-inch feed per tooth.

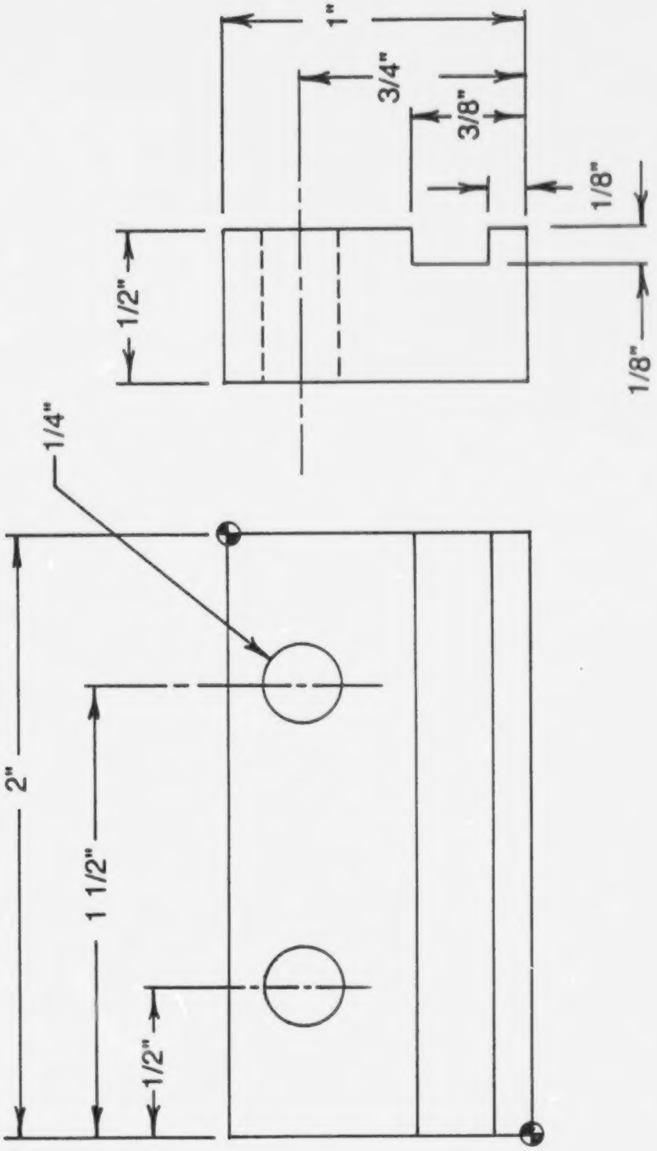
   A. Spindle speed RPM

   B. Feedrate

2. List required operations

   A.

   B.


1"
3/4"
3/8"
1/8"
1/2"
1/8"
1/4"
2"
1 1/2"
1/2"

J. PARTS OF A CNC PROGRAM

A CNC program contains a variety of information and functions.

These include:

- Line numbers or N G codes
- Coordinate locations
- M codes
- Tool data
- Spindle speed data
- Feedrate data
- End-of-block character

1. **Line numbers** tell the controller what line of code is being read. They are written as "N" followed by a number. It is common practice to skip numbers so that additional instructions can be added between lines if necessary later.

N001
N002
N003
etc.

2. **G codes**. G codes, also known as "preparatory functions" come before the coordinates. The G code describes a particular action that will be taken at the X, Y, or Z axis. G codes consist of the letter "G" and a two-digit number.

G00
G01
G99

3. **Coordinate locations.** These are the X, Y, and Z information that tells the controller where to move the tool over the workpiece.

X0.10
Y0.25
Z - 0.10

4. **M codes**. M codes, also known as miscellaneous functions, control non-motion aspects of the machine. These actions might include turning the spindle on and

M05
M30

off, turning coolant on and off, and program control functions such as ending the program.

M codes consist of the Letter "M" followed by a two-digit number.

5. **Tool data** is used to tell machines that have automatic tool changes which tool to get and install in the spindle.

T01
T02
T10

Tool data is designated by a letter "T" and a two-digit number.

6. **Spindle speed data** tells the machine how fast to run the spindle.

   Spindle speed data is designated by the letter "S" followed by the spindle speed in revolutions per minute (RPM) or a code number that stands for a given spindle speed.

   S1000
   S275

7. **Feedrate data** may be expressed in either inches per minute (IPM) or meters per minute (MPM). Feedrate is designated by the letter "F" followed by the feedrate.

   F10.1
   F200

8. **End-of-block character.** Each line of CNC instructions is referred to as

   EOB

a block. An end of block character tells the machine controller that a particular line of instructions has ended. It is now ready to read the next line. The end-of-block character is like a period at the end of a sentence. We designate it as "EOB."

Sample G Codes:

G00 - Rapid travel
G01 - Linear Motion with a specified feedrate
G90 - Absolute input
G91 - Incremental input

Sample M Codes:

M02 - End of program

M03 - Start spindle rotation

M05 - Stop spindle rotation

M06 - Tool change command

SAMPLE CNC PROGRAM

```
N1000M03T01S100$
N1100M08$
N1200G00X.500Y.750$
N1300Z.2$
N1400G01Z-.600F10$
N1500G00Z.2$
N1600X1.500Y.750$
N1700Z.2$
N1800G01Z-.600F10$
N1900G00Z.2$
N1925M05$
N1950M06T02$
N1975M03$
N2000X-.250Y.312$
N2100Z.125$
N2200G01X2.250F10$
N2300G00Z.2$
N2350M09$
N2400M02$
END
```

DO SKILL SHEET 3

USING A CNC MACHINE

SKILL SHEET 3

Part 1. Match the CNC code letters on the left with their purpose on the right.

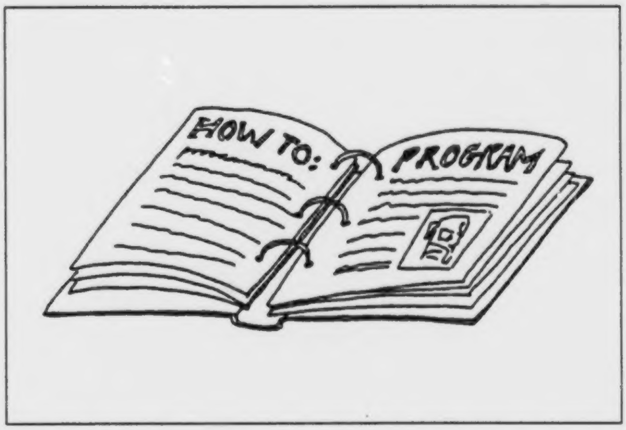
____ 1. N
____ 2. G
____ 3. M
____ 4. T
____ 5. F
____ 6. S

A. Tool number
B. Feedrate
C. Line number
D. Controls machine motion
E. Miscellaneous (non-motion) functions
F. Spindle speed

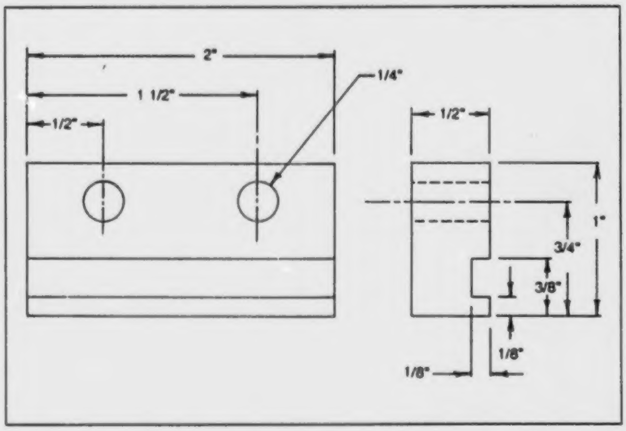
Part 2. Using the sample CNC program on page 32, list G & M codes. Write the definition of each code you find.

K. CONVERTING YOUR PART DRAWING TO A CNC PROGRAM

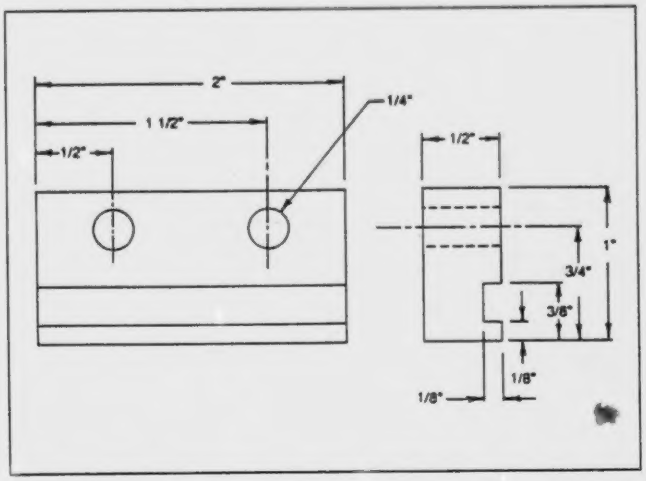
The next step in developing a CNC program is to convert the information on the drawing into a program written on paper. This paper program is sometimes called a manuscript. The completed manuscript contains all the information necessary to enter the program into the machine controller.



On this particular part, we will define the origin point (X0, Y0, Z0) as being in the lower left-hand corner of the part. Z0 is the very top of the part.

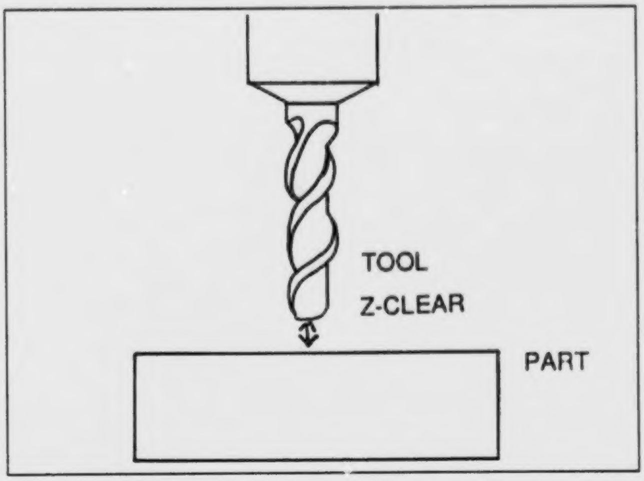


To get started, locate all important part features and give them X, Y, Z coordinate values. The important features on this part are the two holes and the slot. The hole locations are dimensioned from their centers. The important slot dimension is the location of the centerline running through the slot.



LOCATIONS ON PART

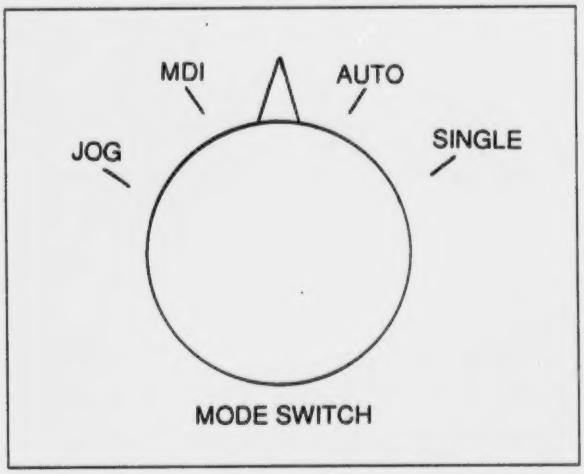
You must also consider raising the tool above the workpiece as it travels over it from one location to the next so that no cutting takes place in the wrong location. We call this "Z-clear." We will establish z-clear as being 0.02 inch above the part.



L. ENTERING CNC DATA TO THE CONTROLLER

Before you enter data from your written program to the CNC controller, you must understand the various operating modes of the controller. Most controllers will have the following modes of operation:

MDI--Manual Data Input
JOG
AUTOMATIC
SINGLE STEP

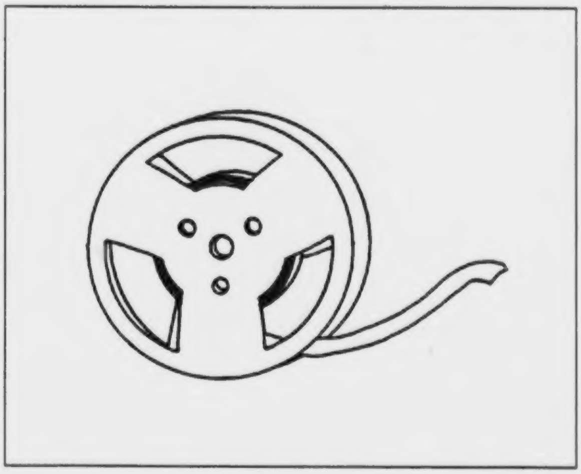


A switch on the front of the controller allows the operator to switch from one mode to the next.

1. MDI allows entry of data through the button on the controller.

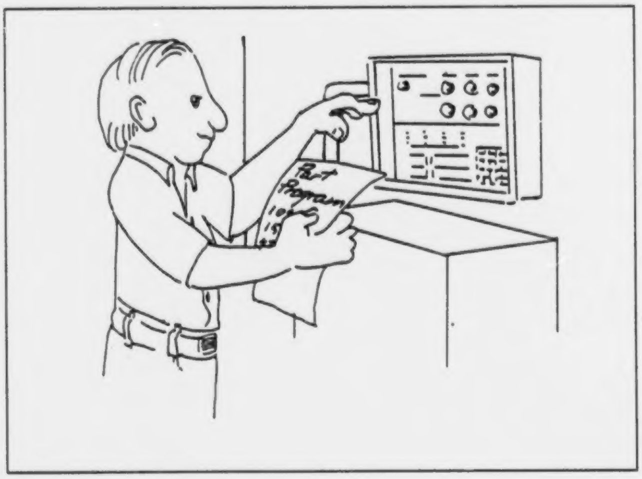
2. JOG mode allows the machine's axes to be positioned (jogged) manually.
3. SINGLE STEP mode permits one block of code to be executed at a time when you push the CYCLE START button.
4. AUTOMATIC mode will execute the entire program when you push the CYCLE START button.

There are several ways to enter CNC data to a machine controller. The most common has been to use a punched paper tape that is read by a tape reader and then transmitted to the controller over a wire.



More recently, many shops are sending the data directly to the controller from a personal computer.

You will enter your program through the control panel of your machine by Manual Data Input (MDI). MDI makes use of the various buttons and switches on the machine and a numeric (numbers) keypad. You enter information from the paper manuscript by hand to the machine controller.



NOTE: Your instructor should review with you safety rules and operating procedures for your particular machine at this time.

CAUTION: Do not proceed until you have received approval from your instructor and you are confident that you can continue safely.

ENTER DATA CAREFULLY

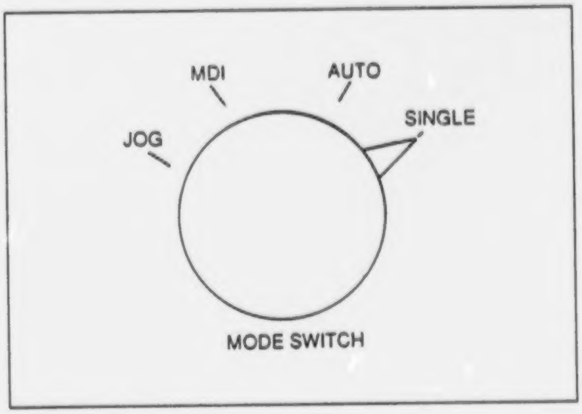
Enter the data exactly as it appears on your revised manuscript. Start with line number (N), and enter all data on that line. End the line with an END-OF-BLOCK character (EDB).

M. TESTING THE PROGRAM

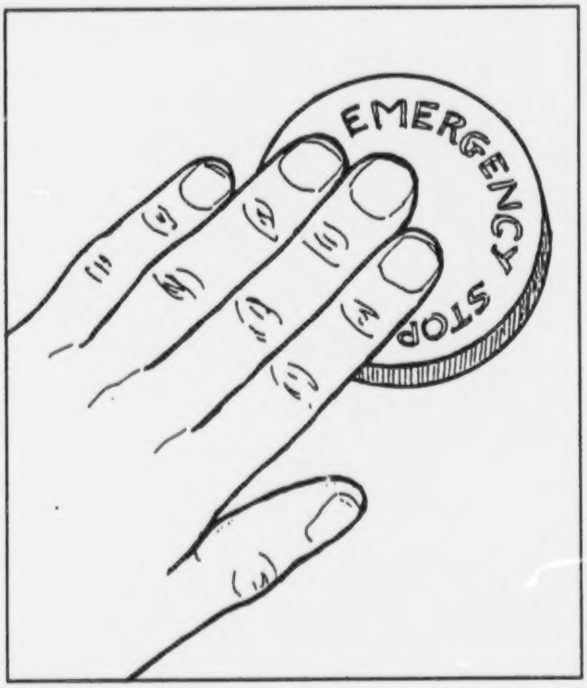
1. To test your program, remove all tooling and workholding devices from the machine. It's important that there is nothing to crash into when you operate the machine for your test.

DO NOT USE ANY TOOLING WHILE TESTING YOUR PROGRAM

2. Set a ZERO POINT somewhere in the center of the machine table. **(YOUR INSTRUCTOR SHOULD WATCH YOU DO THIS!)**



3. Set the machine for single block operation. In this mode the machine pauses after each line of code. You must press the CYCLE START button after each block is executed.



4. While you keep one hand near the EMERGENCY STOP button, press CYCLE START.

5. Observe machine motion and make note of any movements that you did not

expect. Common errors include giving dimensions that are incorrect, giving the wrong sign (=/-) for a dimension, and leaving out information such as "G" or "M" codes.

6. Make changes on your manuscript and then change the program in the controller. Your machine manual should contain information on editing the program in the controller.

7. Repeat the procedure until you are satisfied that the program is correct.

DO SKILL SHEET 4

USING A CNC MACHINE

SKILL SHEET 4

Part 1

1. A CNC program written on paper may be called a________________

2. What does Z-clear mean?

Part 2. Match the four CNC modes of operation with their definitions.

A. MDI

B. AUTOMATIC

C. JOG

D. SINGLE STEP

___ 3. Executes entire program when you push CYCLE START.

___ 4. Allows the machine's axes to be positioned manually.

___ 5. Allows entry of data through the button on the controller.

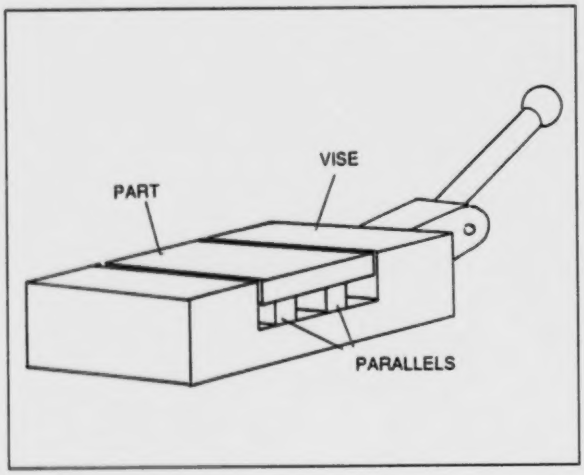
___ 6. Permits one block of code to be executed at a time when you push the CYCLE START button.

Part 3.

7. Why should you NOT use tooling when testing a CNC program?

8. What operating mode should you use to test a program?

N. SETTING UP THE CUTTING TOOL AND WORKPIECE



Setting up the cutting tool and workpiece is not much different on the CNC machine than it is in conventional machining.

1. Mount the vise on the table securely.

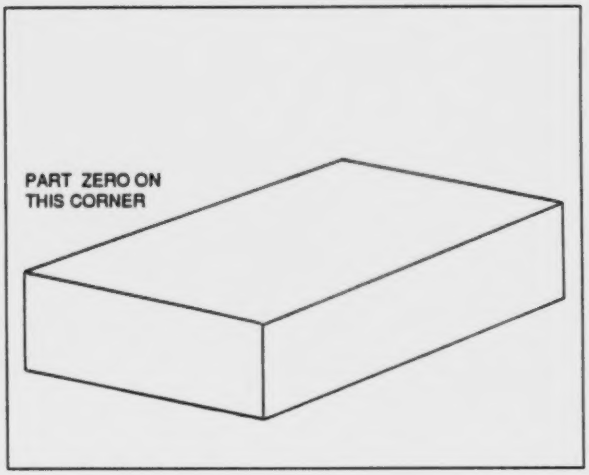
2. Secure the workpiece in the vise. Use parallels to raise the workpiece on vise. This will prevent cutting the vise when you drill the two holes.

3. Mount the endmill in the machine spindle.

4. Have your instructor check your work.

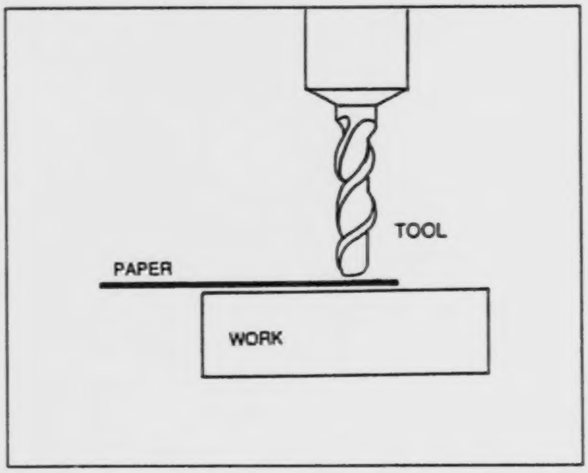
5. Now set ZERO on the machine. Your program zero point and machine zero should be the same when you are done. This will tell the controller where to begin execution of the program. To set ZERO on the machine follow the procedure below:

a. Jog the Z axis up to clear the tool above the workpiece.



b. Jog the X and Y axes until the center of the tool is over the lower left-hand corner of the part. Set X and Y to ZERO. (See machine manual or ask your instructor for help.)

c. Slowly jog the Z axis down until the tool just touches the top of the workpiece. A good way to do this is to bring the tool down close to the work then slide a piece of paper between the tool and the work. When the paper catches and will not slide, you are close enough.



d. Set Z to ZERO.

6. Have your instructor check your work.

7. Raise the Z axis to clear the tool and return all axes to HOME position.

O. MACHINING AND INSPECTING THE PART

NOTE: Your instructor should check your set-up and supervise the machining of your first part.

CAUTION: You should be wearing safety glasses when operating a CNC machine tool.

1. Move the tool to HOME position.
2. Make sure the work area is clear.
3. Set the machine to SINGLE STEP operating mode.

4. With one hand near the EMERGENCY STOP button, press CYCLE START.

5. Continue pressing CYCLE START until the entire program is complete.

   NOTE: If at any time you feel the machine is moving too fast or is going to "crash," stop all motion by pressing EMERGENCY STOP.

6. When the entire program has been executed and the tool has stopped turning, carefully remove the part from the vise.

7. Measure the part to see if it has been made correctly.

DO ACTIVITY SHEET 1

USING A CNC MACHINE

ACTIVITY SHEET 1

Tools and equipment:

Procedure:	Evaluation 3 2 1 0
1. Mount the vise on the table securely.	_ _ _ _
2. Secure the workpiece in the vise. Use parallels to raise the workpiece on vise. This will prevent cutting the vise when you drill the two holes.	_ _ _ _
3. Mount the endmill in the machine spindle.	_ _ _ _
4. Have your instructor check your work.	_ _ _ _

225

5. Now set ZERO on the machine. Your program zero point and machine zero should be the same when you are done. This will tell the controller where to begin execution of the program. — — — —

To set ZERO on the machine follow the procedure below:

A. Jog the Z axis up to clear the tool above the workpiece. — — — —

B. Jog the X and Y axes until the center of the tool is over the lower left-hand corner of the part. — — — —

Set X and Y to ZERO.

(See machine manual or ask your instructor for help.)

C. Slowly jog the Z axis down until the tool just touches the top of the workpiece. A good way to do this is to bring the tool down — — — —

close to the work then slide a piece of paper between the tool and the work. When the paper catches and will not slide, you are close enough.

D. Set Z to ZERO. _ _ _ _

6. Have your instructor check your work. _ _ _ _

7. Raise the Z axis to clear the tool and return all axes to HOME position. _ _ _ _

8. Have instructor check your set-up. _ _ _ _

CAUTION: You should be wearing safety glasses when operating a CNC machine tool.

9. Move the tool to HOME position. _ _ _ _

10. Make sure the work area is clear. _ _ _ _

11. Set the machine to SINGLE STEP operating mode. _ _ _ _

12. With one hand near the EMERGENCY STOP button, press CYCLE START. _ _ _ _

13. Continue pressing CYCLE START until the entire program is complete. _ _ _ _

NOTE: If at any time you feel the machine is moving too fast or is going to "crash," stop all motion by pressing EMERGENCY STOP.

14. When the entire program has been executed and the tool has stopped turning, carefully remove the part from the vise. _ _ _ _

16. Have instructor check your work ____

17. Clean your work area and return tools to their proper locations. ____

USING A CNC MACHINE

UNIT TEST

Part 1. Match the terms below with their definitions.

A. MDI
B. Axis
C. Absolute dimensioning
D. Incremental dimensioning
E. Z-clear
F. Origin
G. Program zero
H. Computer Numerical Control

____ 1. A direction along which a tool or workpiece moves (X, Y, Z)

____ 2. A system in which a computer controls a machine by using commands in a number format

____ 3. Vertical distance above workpiece that will allow the tool to clear while moving from one position to the next

____ 4. Mode for entering data to the controller manually (by hand)

____ 5. A dimensioning system in which each new dimension is taken from the last dimension or machine movement

____ 6. A dimensioning system in which each new dimension is taken with respect to a common origin point

____ 7. The coordinate location with the value X0, Y0, and Z0

____ 8. The reference coordinate from which the part program coordinates are taken

Part 2. Answer each question below.

9. List two advantages of CNC machining over conventional machining.

A.

B.

___10. Which mode permits the operation of one line of a CNC program at a time?

A. MDI

B. JOG

C. SINGLE STEP

D. AUTOMATIC

Part 3. Match each program function to its definition

___11. Indicates line number

___12. Indicates tool number

___13. Miscellaneous function code

___14. Spindle speed

___15. Preparatory function

A. N

B. M

C. G

D. T

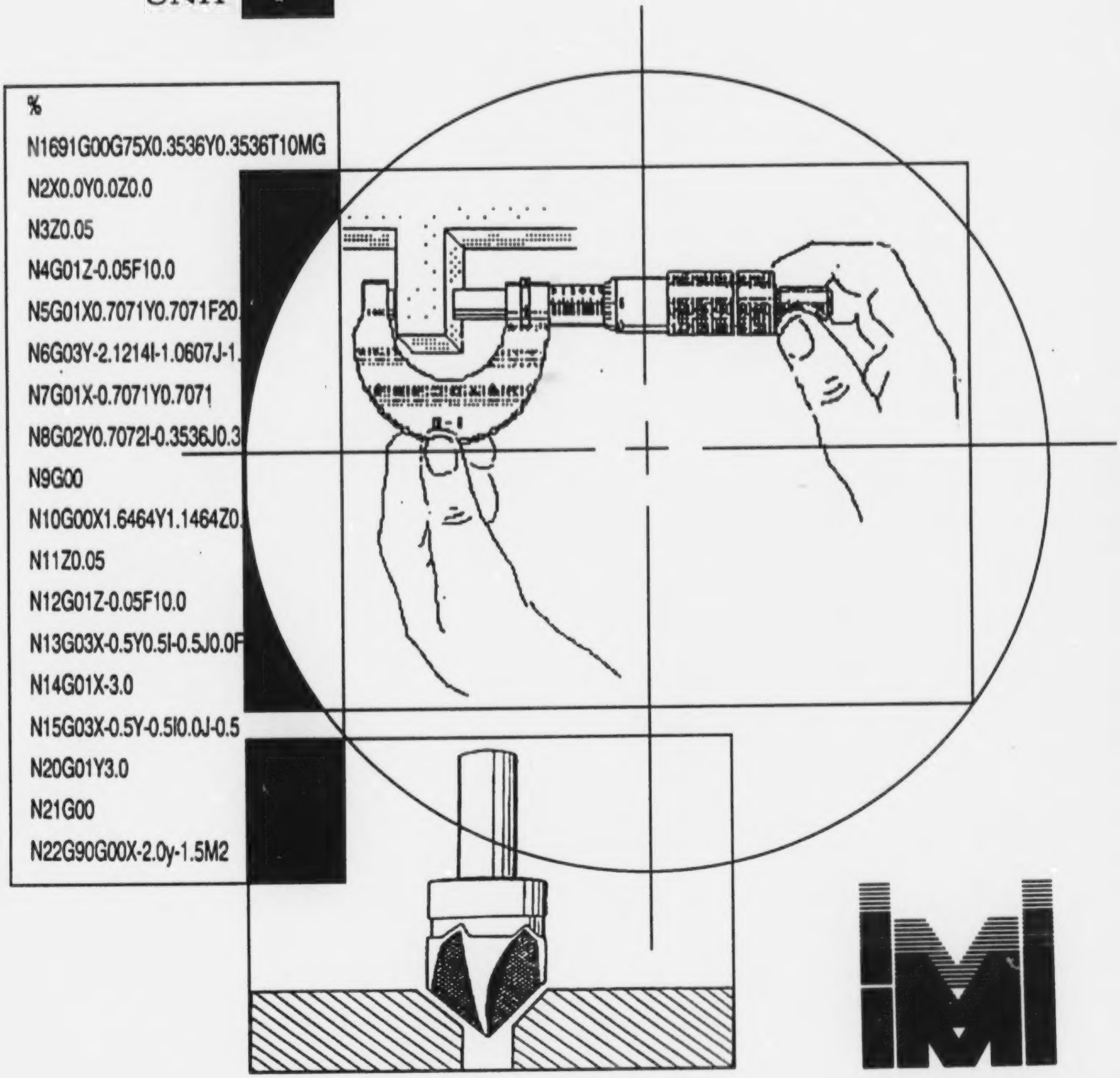
E. S

Machine Shop

90-9200-S
SUPPLEMENTARY UNITS

CNC MACHINE SAFETY

UNIT 7



```
%
N1691G00G75X0.3536Y0.3536T10MG
N2X0.0Y0.0Z0.0
N3Z0.05
N4G01Z-0.05F10.0
N5G01X0.7071Y0.7071F20.
N6G03Y-2.1214I-1.0607J-1.
N7G01X-0.7071Y0.7071
N8G02Y0.7072I-0.3536J0.3
N9G00
N10G00X1.6464Y1.1464Z0.
N11Z0.05
N12G01Z-0.05F10.0
N13G03X-0.5Y0.5I-0.5J0.0F
N14G01X-3.0
N15G03X-0.5Y-0.5I0.0J-0.5
N20G01Y3.0
N21G00
N22G90G00X-2.0y-1.5M2
```

USING A CNC MACHINE

PERFORMANCE CHECKLIST

Student____________________________has successfully performed the following tasks:

	YES	NO	COMMENTS
1. Defined terms associated with CNC			
machine operation.			
2. Identified X, Y & Z axes.			
3. Using a given part, listed the operations that must be performed to complete the part.			
4. Entered data for the part into the CNC controller.			
5. Tested the program.			
6. Set up the cutting tool and workpiece.			
7. Machined and inspected the part.			

Satisfactory - Should move on ☐

Repeat this unit ☐

____
Student's Signature/Date

____
Instructor's Signature/Date

CNC MACHINE SAFETY

INTRODUCTION

This unit will teach you safety rules for using computer numerical control (CNC) machine tools. Of course, all general safety rules of the machine shop also apply to using CNC machine tools. CNC safety is important for two reasons: 1. The machine is moving under computer control, and the operator may not always know what the machine will do next. 2. The equipment is often expensive to repair if damage should occur.

UNIT OBJECTIVE

After completing this unit, you should know the parts of a CNC machine and how to use it safely.

SPECIFIC OBJECTIVES

After completing this unit, you should be able to:

1. Define terms associated with a CNC machine.
2. Identify parts of a CNC machine.
3. List rules for using a CNC machine safely.

Put a check beside the statement that is true for you.

☐ I can do these tasks and want to go through the Performance Checklist.

SEE YOUR INSTRUCTOR

☐ I want to study the information in this unit before I do the Performance Checklist.

TURN PAGE AND BEGIN

A. TERMS

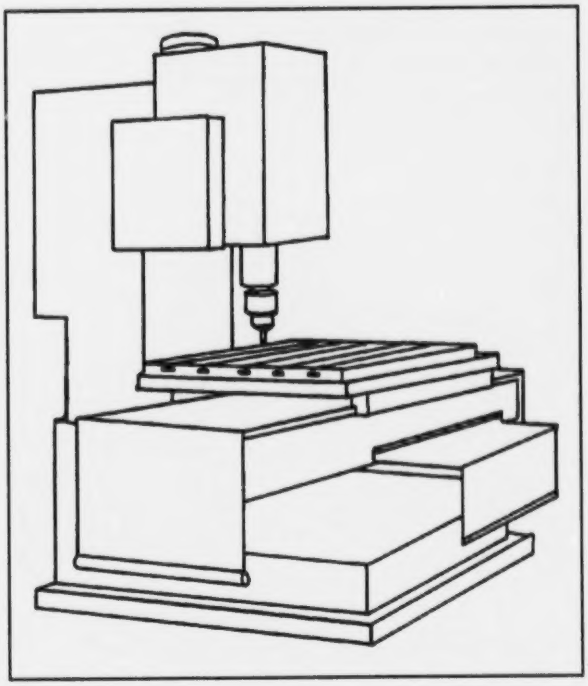
1. **Computer numerical Control (CNC)** - control of machine tool movement and operation by a computer or machine controller.
2. **Feed override** - switch that allows the operator to adjust the feedrate up or down from the computer programmed value during the machine cycle.
3. **Machine Controller** - the computer that controls the operation of a numerical control machine tool.
4. **Automatic Operation** - CNC operating mode in the machine is under control of the machine controller.
5. **Servo-motor** - an electric motor that can drive an axis of the CNC machine tool.
6. **Program** - coded instructions that are read by the machine controller.
7. **Emergency stop button** - usually a red "mushroom" switch. When you press it, it removes all power from the CNC machine tool.
8. **Cycle start button** - button pressed to begin automatic operation of the CNC machine tool.

9. **Dry run** - a test run of machine movements done without cutting tools or a workpiece in place.

10. **Test part** - single part made to test the CNC program to see if it is performing properly.

B. THE IMPORTANCE OF CNC SAFETY

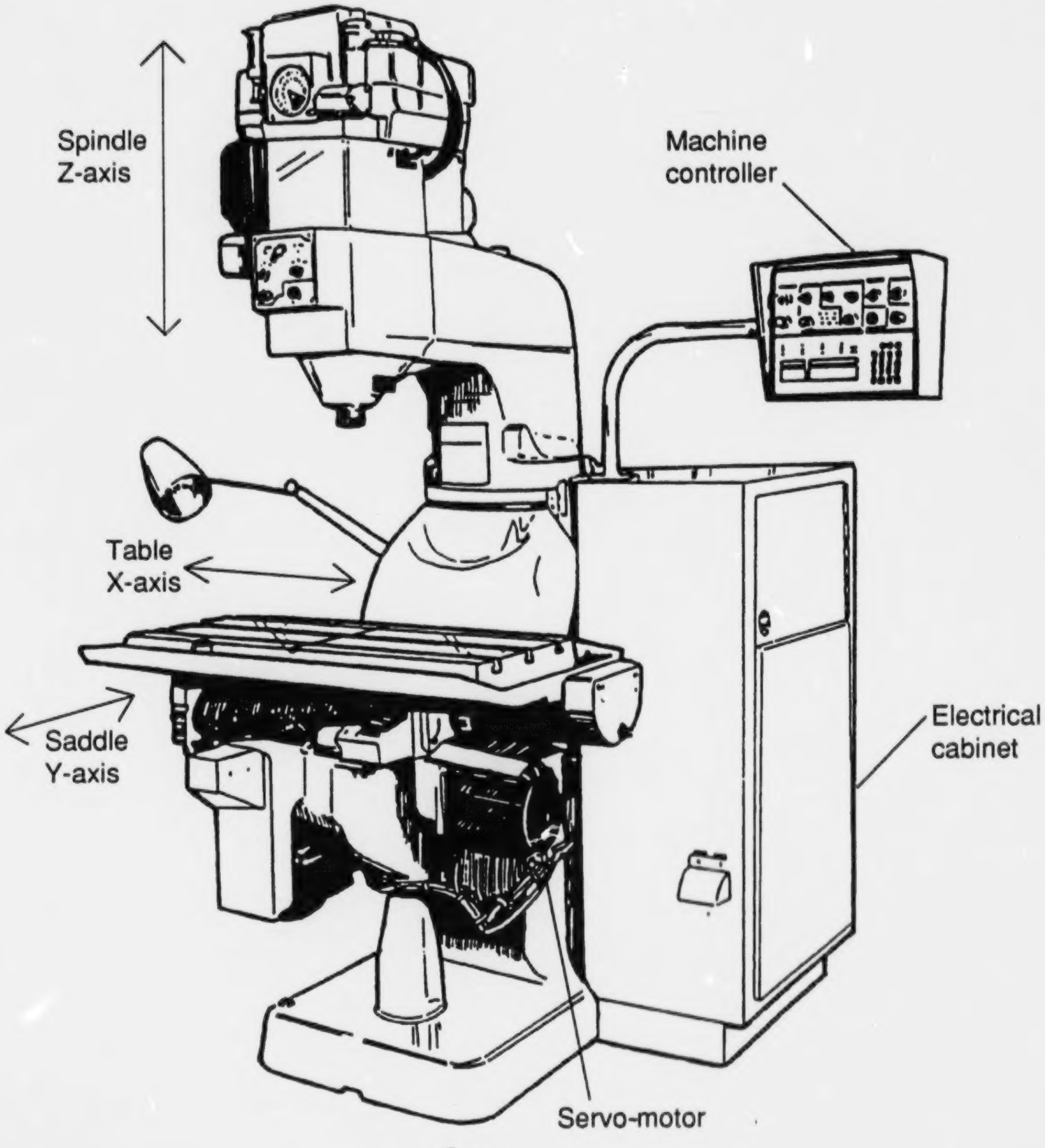
Safety is always very important. You can hurt yourself or someone else really seriously when you work around machines. Safety is especially important to the operation of a CNC machine for two reasons:



1. The machine is moving under computer control, and the operator may not always know what the machine will do next.

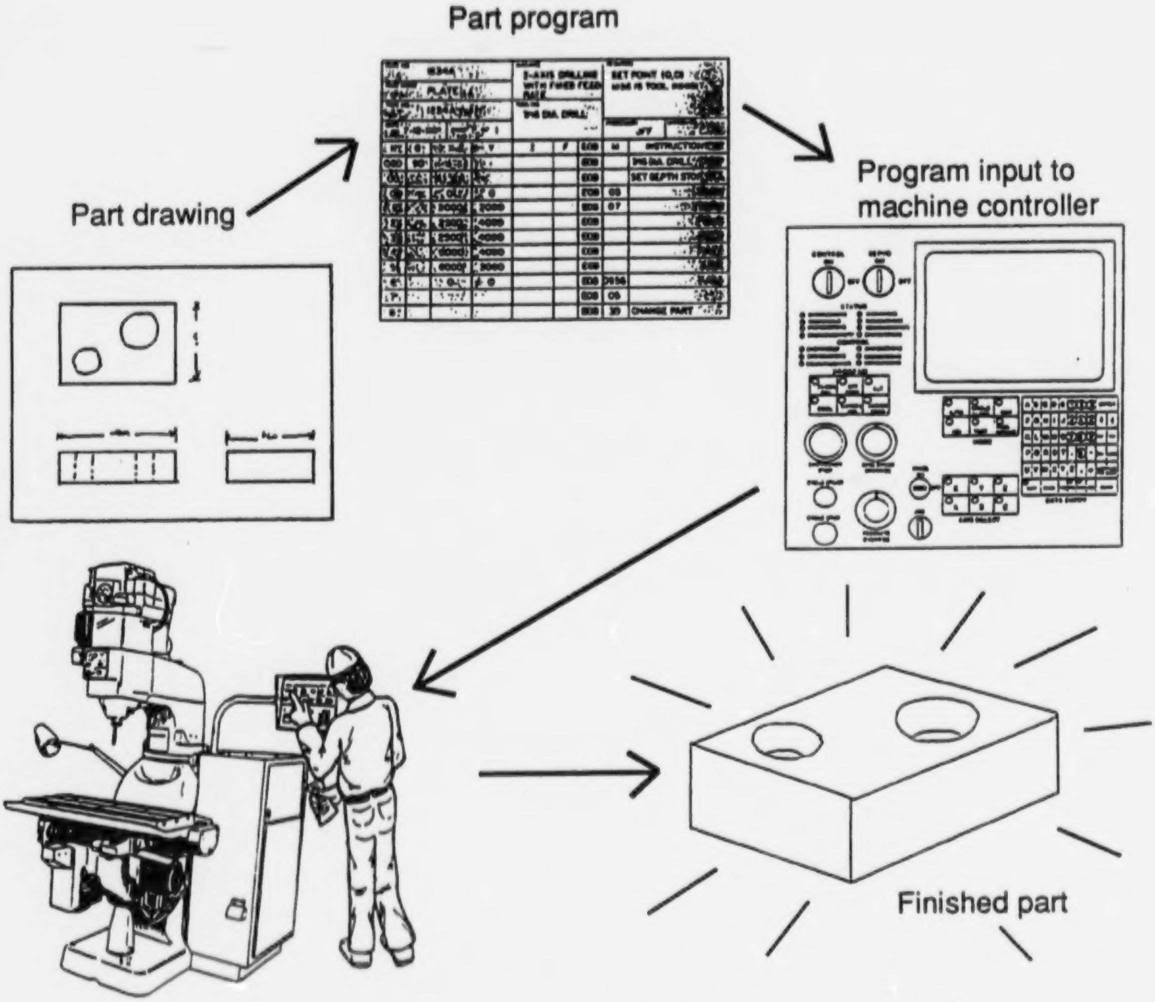
2. The equipment is often expensive to repair if damage should occur.

C. BASIC PARTS OF THE CNC MACHINE TOOL


Spindle
Z-axis
Machine
controller
Table
X-axis
Electrical
cabinet
Saddle
Y-axis
Servo-motor

239

C. SCHEMATIC OF CNC OPERATING PRINCIPLE


Part program
Part drawing
Program input to machine controller
Finished part

DO SKILL SHEET 1

DO ACTIVITY SHEET 1

CNC MACHINE SAFETY

SKILL SHEET 1

Match the CNC terms below with their definitions.

A. Program
B. Servo-motor
C. Dry run
D. Cycle start button
E. Computer numerical control
F. Test part
G. Feed override
H. Machine controller
I. Emergency stop button

___ 1. Control of machine tool movement and operation by a computer or machine controller
___ 2. Switch that allows the operator to adjust the feedrate up or own from the computer programmed value during the machine cycle
___ 3. The computer that controls the operation of a numerical control machine tool
___ 4. An electric motor that can drive an axis of the CNC machine tool
___ 5. Coded instructions that are read by the machine controller
___ 6. Usually a red "mushroom" switch. When you press it, it removes all power from the CNC machine tool.
___ 7. Button pressed to begin automatic operation of the CNC machine tool
___ 8. A test run of machine movements done without cutting tools or a workpiece in place

___ 9. Single part made to test the CNC program to see if it is performing properly

CNC MACHINE SAFETY

ACTIVITY SHEET 1

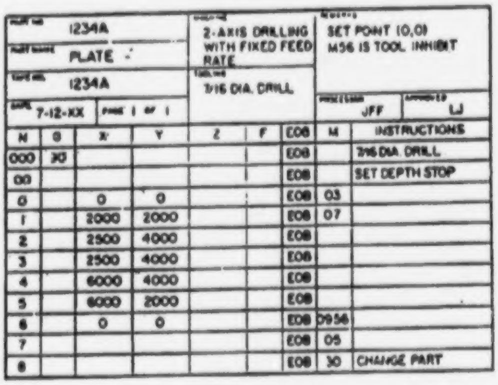
On the CNC machine in your shop, identify the following parts:

1. Machine controller
2. Servo-motor
3. Cycle start button
4. Emergency stop button
5. Feedrate override switch

E. SAFE OPERATION OF CNC MACHINE TOOLS

1. Program control

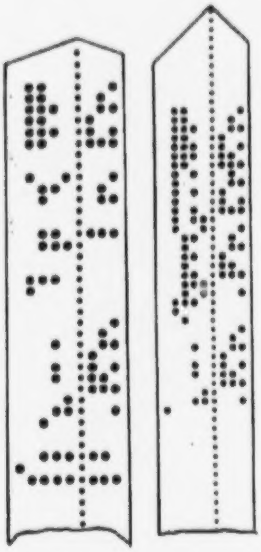
A numerical program (NP) controls the movements of a CNC machine tool. This program is a code that the machine controller reads. The code provides instructions for moving the machine table, turning the machine spindle on and off, etc.



1234A
PLATE
1234A
7-12-XX
PAGE 1 OF 1
2-AXIS DRILLING WITH FIXED FEED RATE
3/16 DIA. DRILL
SET POINT (0,0)
M56 IS TOOL INHIBIT
JFF
LJ

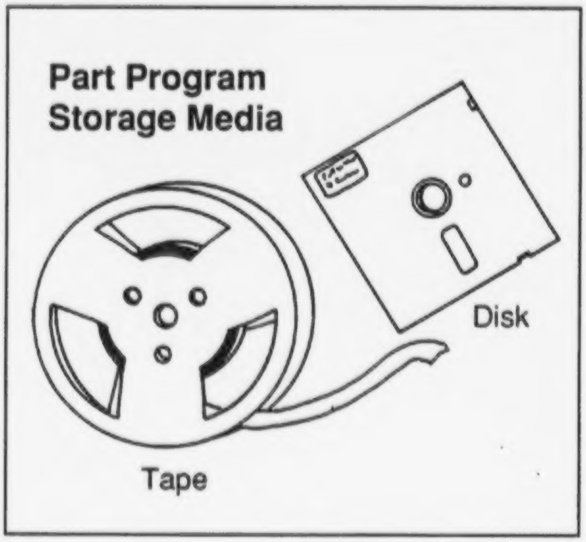
N	G	X	Y	Z	F	EOB	M	INSTRUCTIONS
000	90					EOB		3/16 DIA. DRILL
00						EOB		SET DEPTH STOP
0		0	0			EOB	03	
1		2000	2000			EOB	07	
2		2500	4000			EOB		
3		2500	4000			EOB		
4		6000	4000			EOB		
5		6000	2000			EOB		
6		0	0			EOB	0956	
7						EOB	05	
8						EOB	30	CHANGE PART

Part program



In many cases, a machine operator will be asked to run a program to produce a part that was written by a part programmer. The operator may not be aware of the contents of the program. An operator should meet with the programmer to find out what the programs is intended to do.

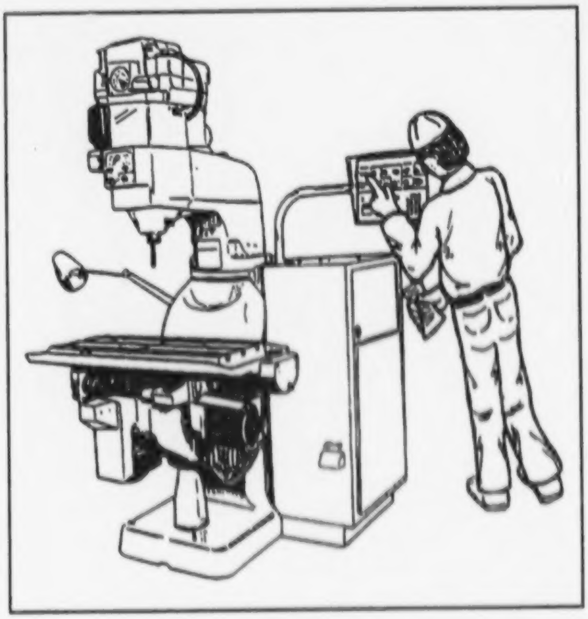
Be sure you know the contents of the NC program that you will run. Ask yourself, "What is the program supposed to do?" "What is the proper order for the various operations in the program?"



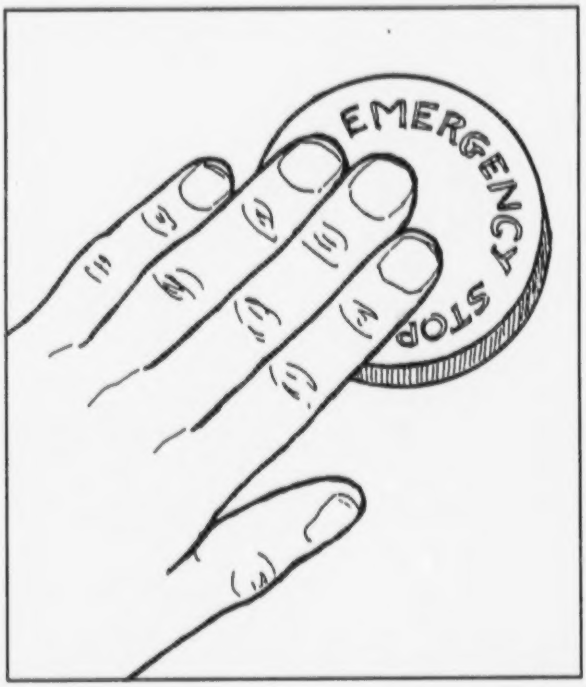
CAUTION: Never operate a CNC machine on which you have not had thorough training.

2. **Doing the dry run**

The next step is to perform a dry run. In a dry run the machine goes through all the motions of the program. The machine contains no tooling and there is no workpiece.



Remove all cutting tools and workholding devices (such as vises and clamps) from the machine. Many machines have feedrate override. If yours does, you may use it during the dry run to lower the feedrate so that you can have better control of the machine motions and stop them if necessary. Keep one hand on the emergency stop button and press cycle start.



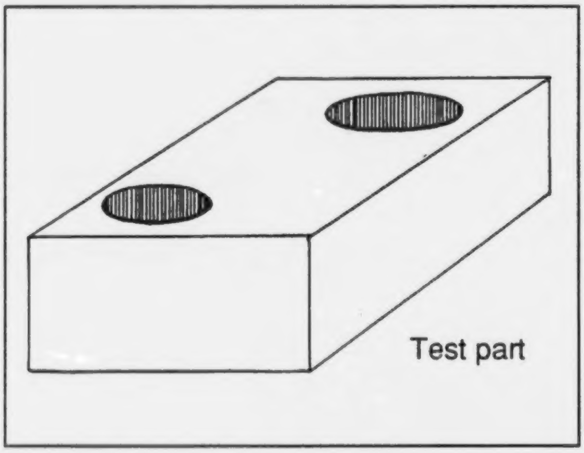
246

NOTE: If your machine has the ability to step through the part program one block at a time, you should use this feature during the dry run.

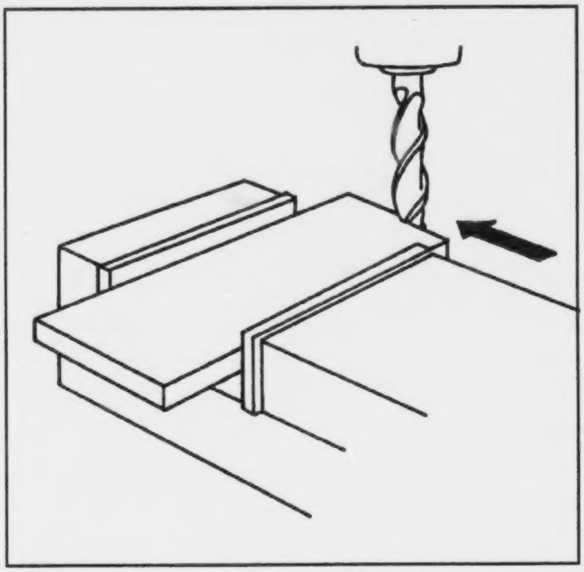
Watch the machine motions and sequence of operations. Make sure they are what you want. If they are not, the program should be modified and tested again.

3. **Making the test part**

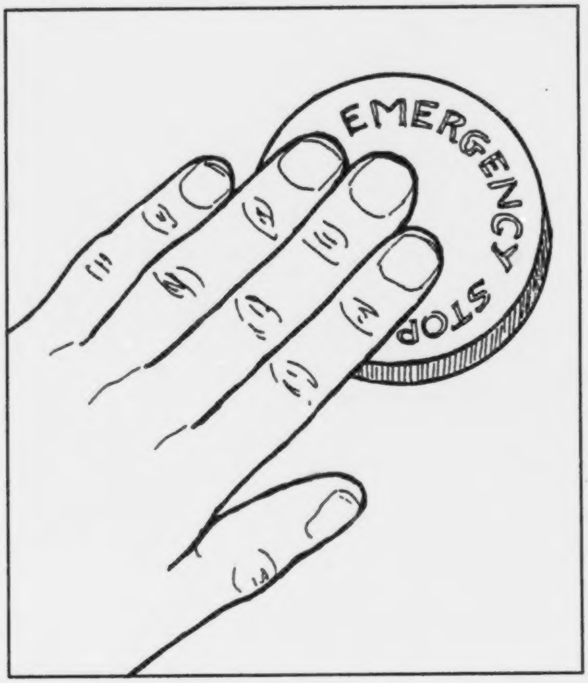
After you have successfully completed the dry run, it is time to make a test part. Get a suitable piece of workpiece material and the necessary cutting tools and vise or clamps.



NOTE: Often a test part will be made from machinable wax or other soft material. This is especially true if the actual workpiece material is extremely expensive.

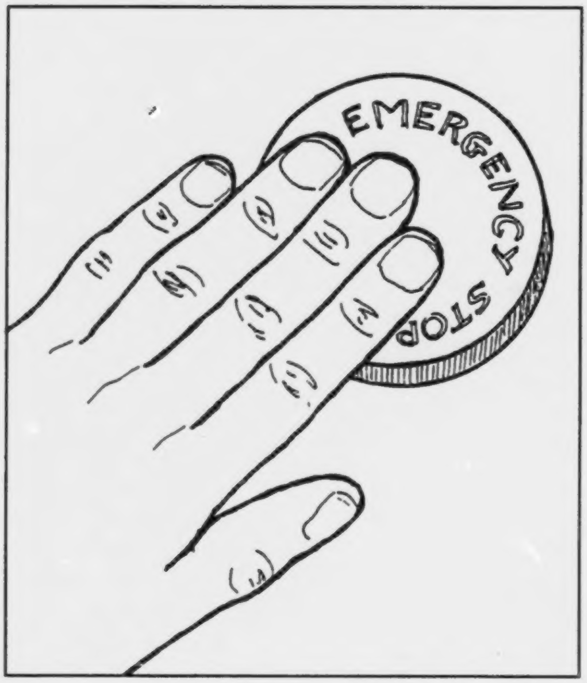


Mount the workpiece and tools. Perform machining as prescribed using the feedrate override and single-step operation (if that is available). Be prepared to press the emergency stop button at all times in case it becomes necessary to stop the machine motion quickly to avoid a "crash."



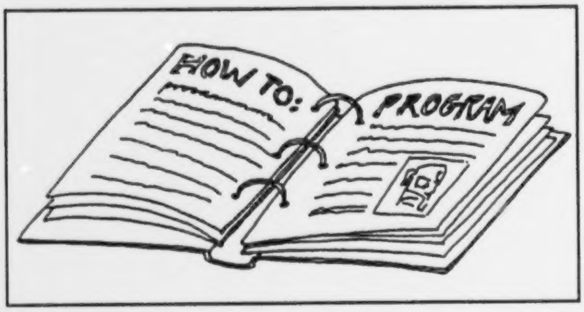
F. SPECIAL SAFETY RULES FOR USING A CNC MACHINE

When you operate a CNC machine, all general safety rules concerning machine tools and shop work apply. The rules below are specific to CNC machines.



1. Do not operate controls unless you have been instructed in their use. Remember, The cycle stop button stops the machine as soon as the command has been performed. The emergency stop button halts the feed of all axes immediately.

2. Never assume. Ask about anything you are unfamiliar with.



3. Do not attempt to program a machine without programming manuals.

4. Maintain a safe space above the workpiece.

5. Keep your hands away from the rotating spindle and moving turrets.

6. Know where the tool is going AT ALL TIMES.

DO SKILL SHEET 2

CNC MACHINE SAFETY

SKILL SHEET 2

Fill in the blanks

1. Movements of the CNC machine tool are controlled by________________________________.
2. The CNC program is read by the______________________.
3. During a dry run, you should keep one hand on the____________ at all times.
4. List four safety rules that apply to CNC machining.
   A.
   B.
   C.
   D.

CNC MACHINE SAFETY

UNIT TEST

Part 1. Directions: Place a "T" in front of statements that are true. Place an "F" in front of statements that are false.

___ 1. In CNC machining, the machine is moving under the control of the computer.

___ 2. CNC machining equipment is easy to repair and is usually quite inexpensive.

___ 3. The feed override switch allows the operator to adjust the feedrate up or down from the programmed value during the machining cycle.

___ 4. The "machine controller" is the operator who runs the machine.

___ 5. In automatic operating mode, the machine is under control of the computer.

___ 6. The "program" in CNC operation is the same as the part blueprint.

___ 7. The emergency stop button will remove power from the machine.

___ 8. A dry run is so called because no cutting fluid is used in making a test part.

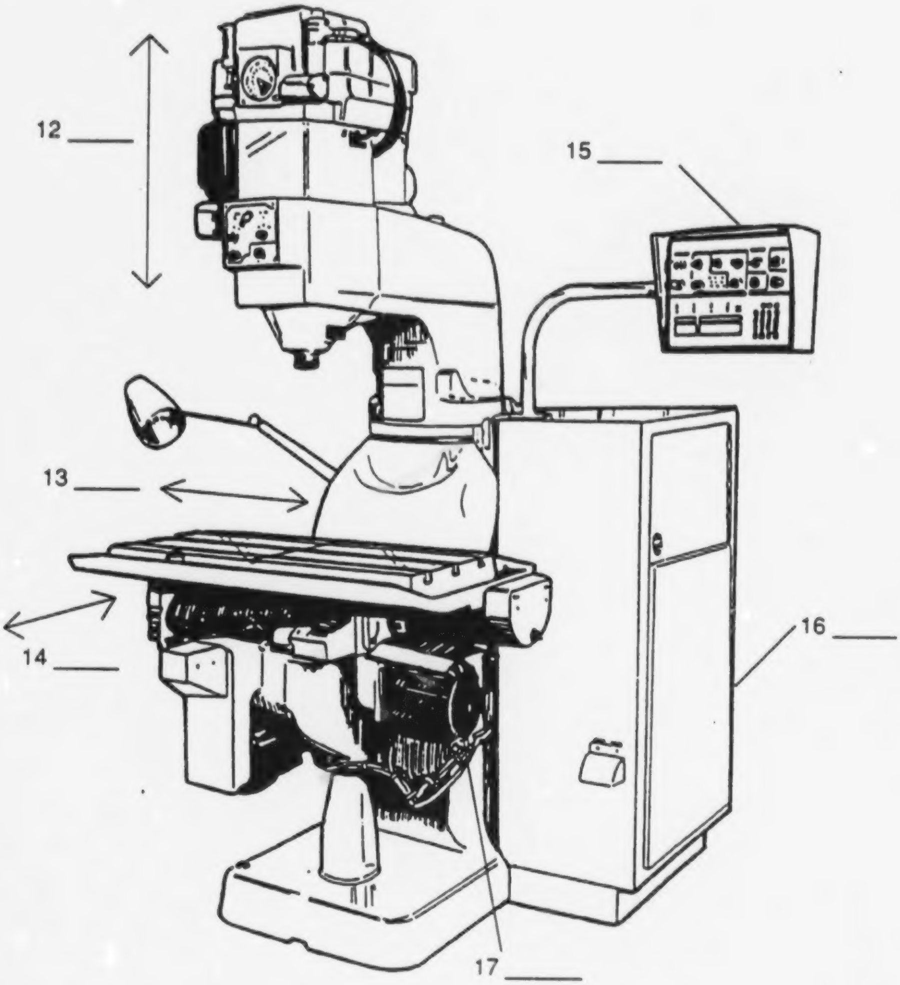
___ 9. The movements of a CNC machine tool are determined by the contents of the CNC program.

___10. The purpose of making a test part is to watch for crashes and check the accuracy of the part after it is made.

___11. CNC machine tools are so different from conventional machines that few, if any, standard safety rules apply.

Part 2. Match the part names below with their correct location on the CNC milling machine.

A.	Saddle (Y-axis)	D.	Spindle (Z-axis)
B.	Electrical cabinet	E.	Table (X-axis)
C.	Servomotor	F.	Machine controller



CNC MACHINE SAFETY

PERFORMANCE CHECKLIST

Student____________________________has successfully performed the following tasks:

	YES	NO	COMMENTS
1. Defined terms associated with a CNC machine.			
2. Identified parts of a CNC machine.			
3. Listed rules for using a CNC machine safely.			

Satisfactory - Should move on ☐

Repeat this unit ☐

____
Student's Signature/Date

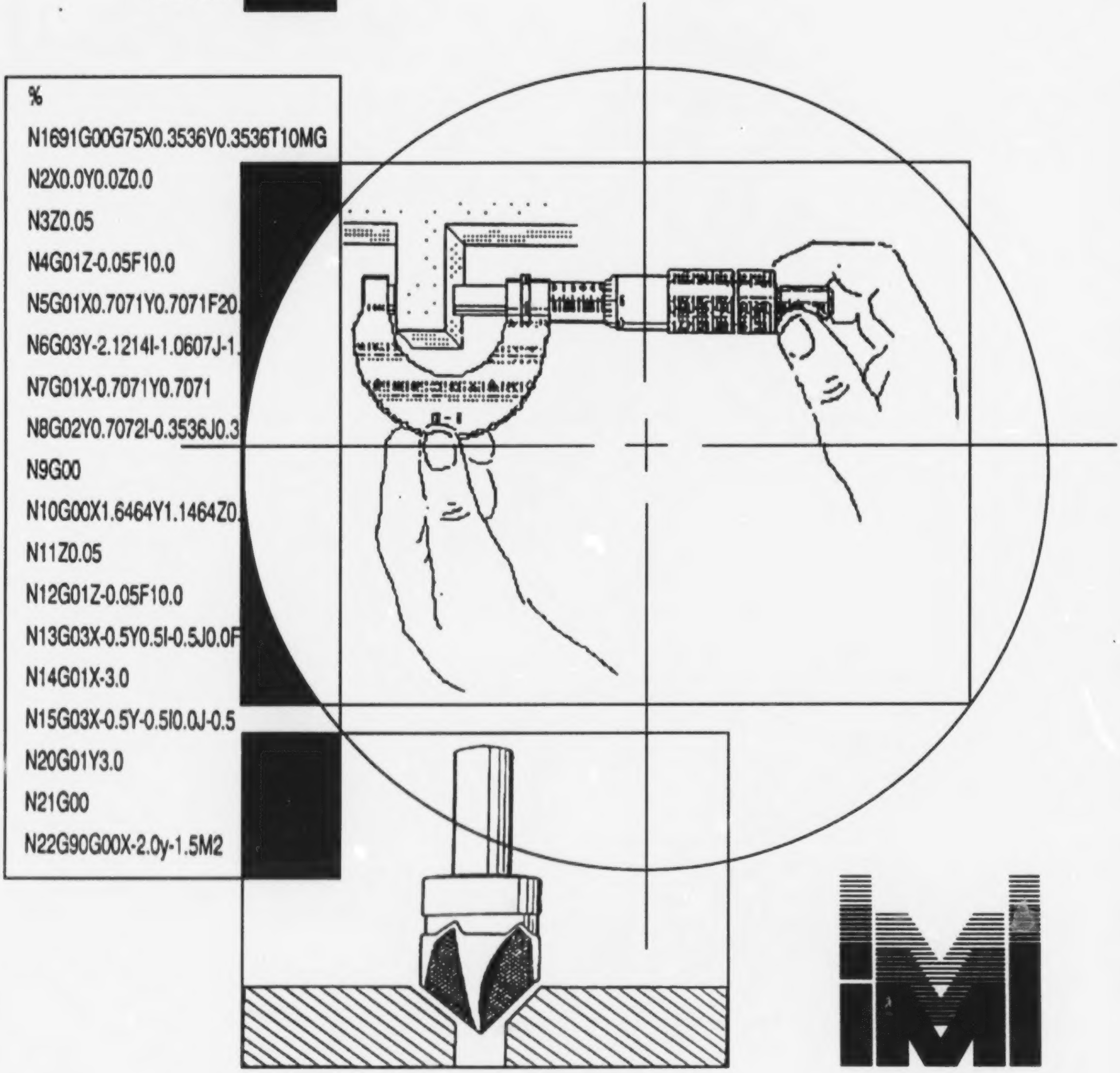
____
Instructor's Signature/Date

Machine Shop

90-9200-S
SUPPLEMENTARY UNITS

TOOL VARIABLES FOR MACHINING

UNIT 8



```
%
N1691G00G75X0.3536Y0.3536T10MG
N2X0.0Y0.0Z0.0
N3Z0.05
N4G01Z-0.05F10.0
N5G01X0.7071Y0.7071F20
N6G03Y-2.1214I-1.0607J-1
N7G01X-0.7071Y0.7071
N8G02Y0.7072I-0.3536J0.3
N9G00
N10G00X1.6464Y1.1464Z0
N11Z0.05
N12G01Z-0.05F10.0
N13G03X-0.5Y0.5I-0.5J0.0F
N14G01X-3.0
N15G03X-0.5Y-0.5I0.0J-0.5
N20G01Y3.0
N21G00
N22G90G00X-2.0y-1.5M2
```

IMI

TOOL VARIABLES FOR MACHINING

INTRODUCTION

One of the machinist's most important jobs is identifying and determining the tool variables necessary to produce good results. These variables include tool materials and types, cutting fluids, machine conditions and workpiece materials. (Speeds and feeds are important tool variables. They are covered in another section. This information is not repeated in this unit.)

UNIT OBJECTIVE

After completing this unit, you should be able to consider tool variables and how they influence your work.

SPECIFIC OBJECTIVES

After completing this unit, you should be able to:

1. Identify and select tools based upon workpiece material and machine conditions.
2. Match cutting fluid types to their uses.

Put a check beside the statement that is true for you.

☐ I can do these tasks and want to go through the Performance Checklist.

SEE YOUR INSTRUCTOR

☐ I want to study the information in this unit before I do the Performance Checklist.

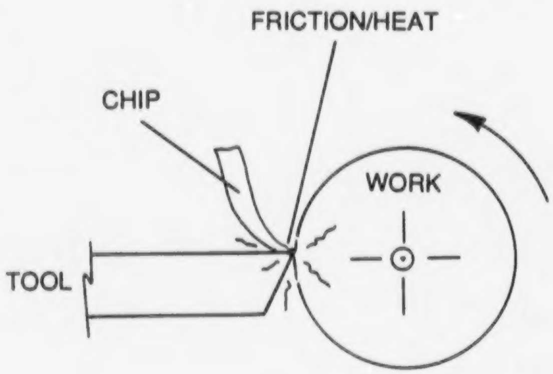
TURN PAGE AND BEGIN

A. TERMS

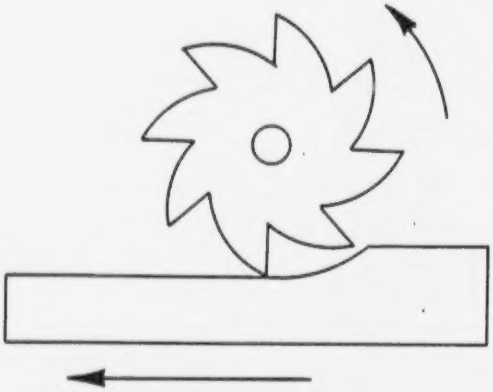
1. **Climb cutting (down milling)** - a milling cutting action in which the cutter turns in the same direction as the workpiece is moving.
2. **Conventional milling (up milling)** - a milling cutting action in which the cutter turns in the opposite direction from the workpiece motion.
3. **Gibs** - thin pieces of metal between mating parts that are adjusted to take up normal wear on machine.
4. **Red hardness** - a characteristic of a cutting tool (especially high-speed steel) that permits it to be operated at temperatures that turn the tool a dull red color.

B. ACTION OF THE CUTTING TOOL

A cutting tool removes material by tearing or shearing it away from the workpiece. Friction (rubbing of the tool and part together) creates heat at the point where the tool touches the workpiece.

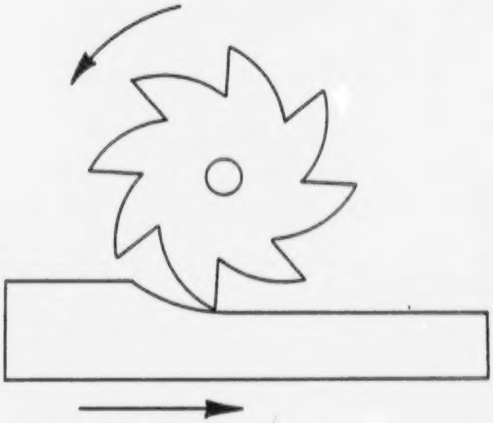


In milling, the tool turns, and the workpiece moves under it. If the workpiece moves in the same direction as the tool is turning, we call the cutting action down milling or "climb cutting."



CONVENTIONAL MILLING

If the work moves against the direction of the tool, we call the action conventional milling or up milling. Up milling is used for most cuts.



CLIMB MILLING

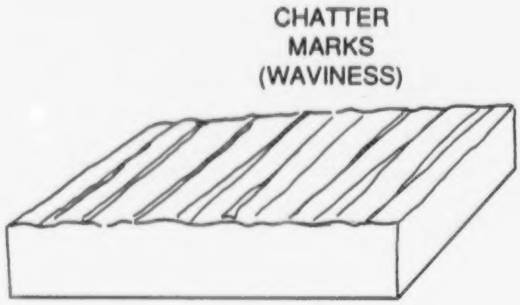
Down milling can cause the tool to dig into the workpiece. Down milling is only used for light, finishing cuts or when you need to have the downward motion of the tool force the work down into the vise.

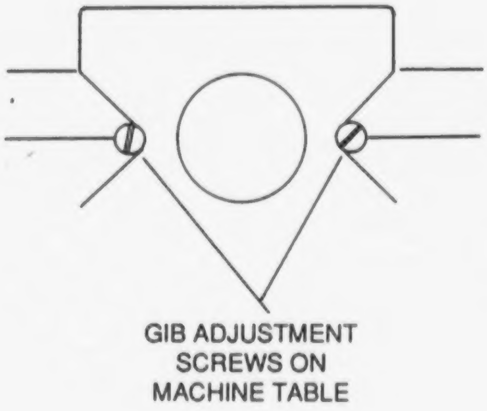
C. MACHINE FACTORS THAT AFFECT CUTTING ACTION

1. **Rigidity of the machine.**

A new machine is "tight." It has little vibration. A tight machine will produce accurate parts with good finish under a wide range of operating conditions.

An older machine, however, may vibrate more. It may produce parts with tool "chatter" marks. Your instructor can adjust the gibs

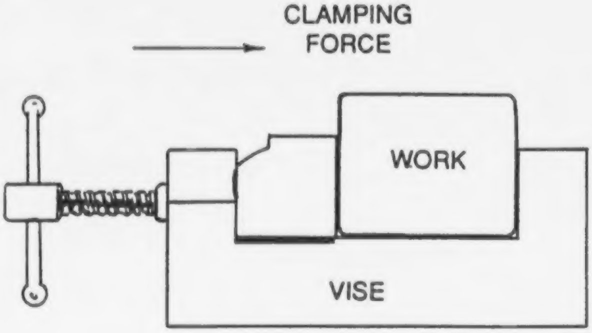

CHATTER MARKS (WAVINESS)


GIB ADJUSTMENT SCREWS ON MACHINE TABLE

and replace the spindle bearings to make a machine tighter.

2. **RIGIDITY OF THE SET-UP**

Some parts require only light clamping pressure to prevent damage. Or the part may have a shape that makes it difficult to clamp.



NOTE: If the cutting force is greater than the clamping pressure, the part will move in the vise. This can cause damage to the workpiece and the tool. Heavy cuts require heavy clamping pressure.

DO SKILL SHEET 1

TOOL VARIABLES FOR MACHINING

SKILL SHEET 1

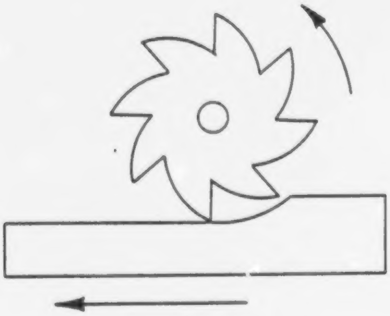
Choose the best response to complete each statement.

___ 1. The action of the cutting tool touching the workpiece produces_____, which creates heat.

A. dampness C. noise

B. friction D. fumes

___ 2.



In this illustration the tool is turning in the same direction as the workpiece is moving. This is called___.

A. down milling B. up milling

___ 3. Adjusting the___ can take up some of the wear on a machine.

A. gibs C. friction

B. dampness D. heat

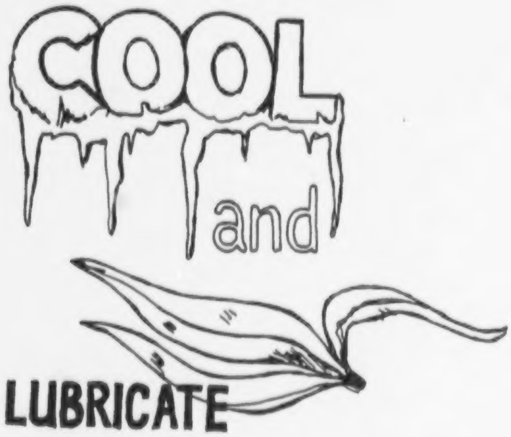
D. CUTTING FLUIDS

Cutting fluids serve the following purposes:

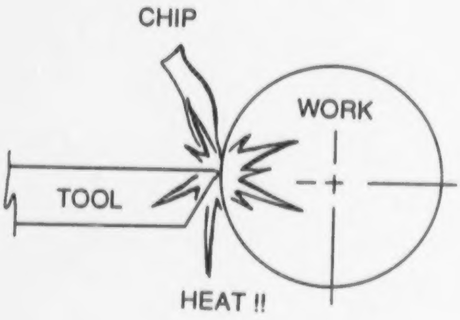
1. Increase tool life
2. Increase the speed of material removal
3. Improve surface finish
4. Improve accuracy

CUTTING FLUID
CUTTING FLUID
CUTTING FLUID

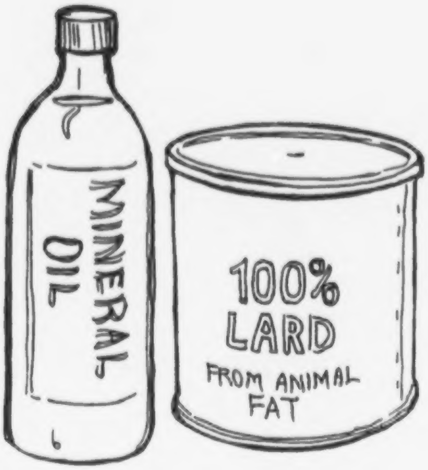
Cutting fluids cool the workpiece and the tool. These fluids also lubricate the face of the tool and allow the chips to slide over it freely.


COOL
and
LUBRICATE

As much as 90 percent of the energy used in machining becomes heat. Cutting fluids can carry this heat away from the tool/work contact point.


CHIP
WORK
TOOL
HEAT !!

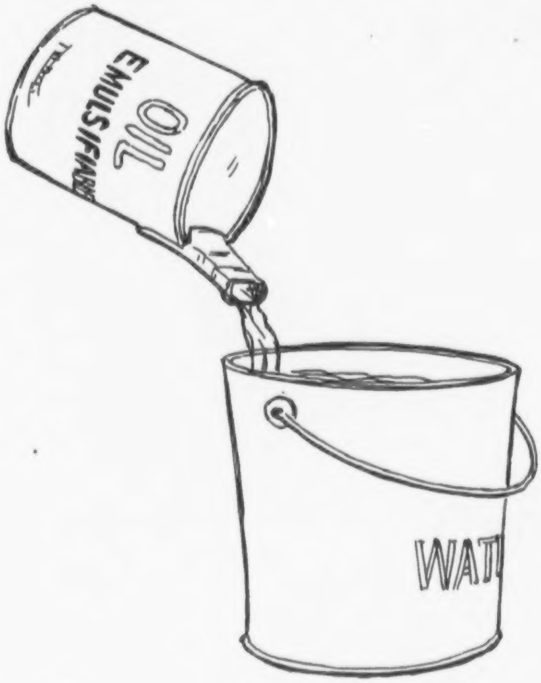
The lubricating action of cutting fluids increase a tool's life by preventing chips from becoming welded to the tool.


MINERAL OIL
100% LARD
FROM ANIMAL FAT

Types of cutting fluids:

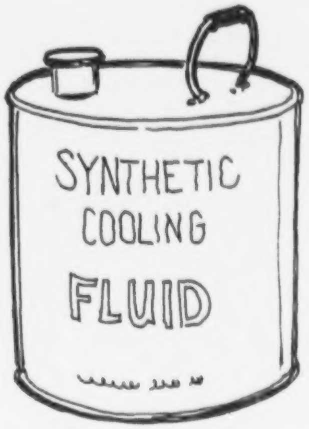
1. **Straight cutting oils**--used for tapping, reaming and machining at high forces. Includes: animal fats (lard) and mineral oils.

2. **Water soluble (emulsifiable) oils**--diluted with water and used to cool and lubricate in turning, grinding and milling


OIL
EMULSIFIABLE
WAT

EMULSIFIABLE OILS

3. **Chemical (synthetic) cutting fluids**--consist of mixture of water and chemical cutting fluids. Used especially for cooling. The chemical keeps the tools, machine and workpiece from rusting as they might if you used just water.



DO SKILL SHEET 2

TOOL VARIABLES FOR MACHINING

SKILL SHEET 2

Choose the best response to complete each statement.

___ 1. Which is not an important purpose of cutting fluids?

A. increase tool life
C. improve surface finish
B. improve accuracy
D. lubricate the machine

___ 2. Cutting fluids work by removing___.

A. heat
C. noise
B. dirt
D. fumes

Place an X beside the three types of cutting fluids.

___ mineral oil
___ vinegar
___ hydrochloric acid
___ synthetic
___ emulsifiable

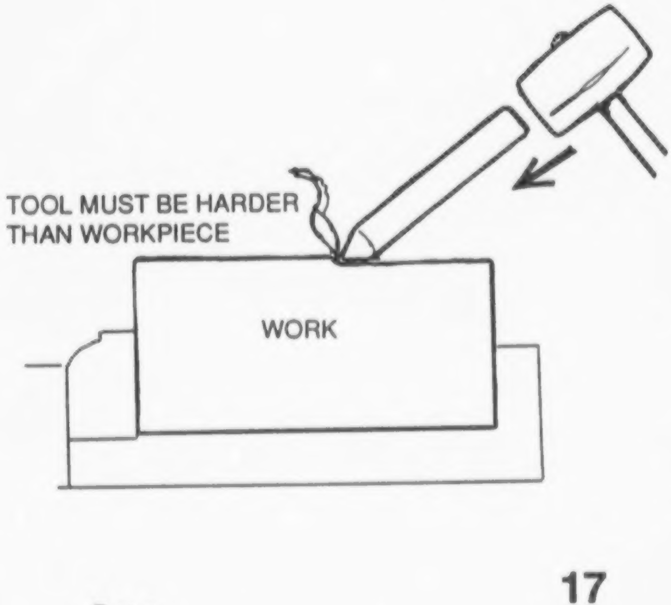
E. TOOL MATERIALS

Selecting tools made from the right material for each job is important. Only with the right tool can you produce good work.

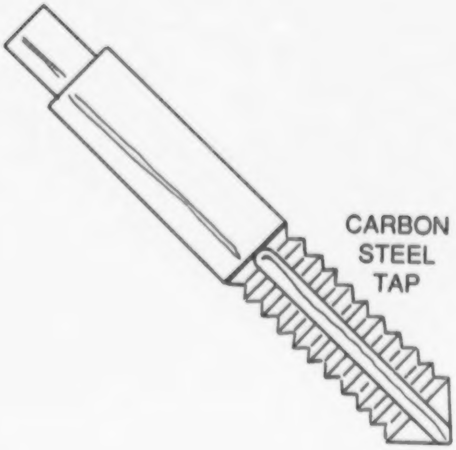
Typical tool materials include:

1. Carbon tool steels
2. High-speed steels
3. Cast alloys
4. Cemented or insert carbide
5. Coated carbides
6. Ceramic
7. Diamond

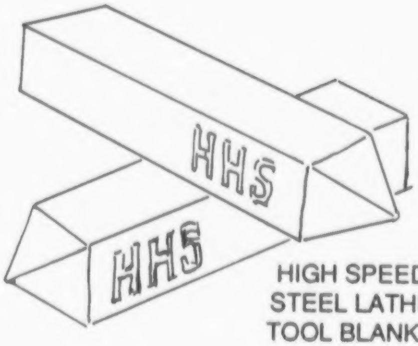
The most important factor in choosing a cutting tool is that the tool must be harder than the workpiece. Otherwise the tool's cutting edge will be worn away quickly.


TOOL MUST BE HARDER THAN WORKPIECE
WORK

1. **Carbon tool steels**--
These are made from high carbon, low alloy steels. They are low in cost, but you cannot operate them a high speeds because the heat created by cutting will cause the tool to lose its hardness. Carbon tool steel tools should be operated at 1/3 to 1/2 the recommended speed for high-speed steel.

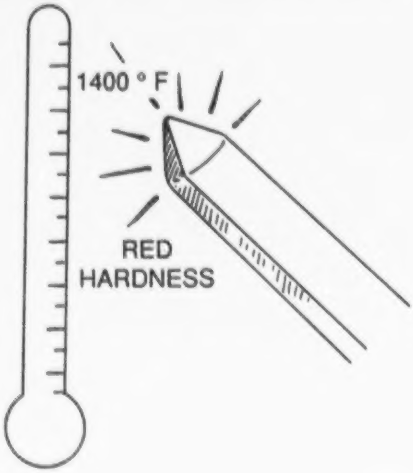

CARBON
STEEL
TAP

2. **High-speed steels**--possess "red-hardness" property that can withstand temperatures that will turn the tool edge a dull red without losing the tool's hardness. They can be operated at much higher speeds than carbon steel tools.

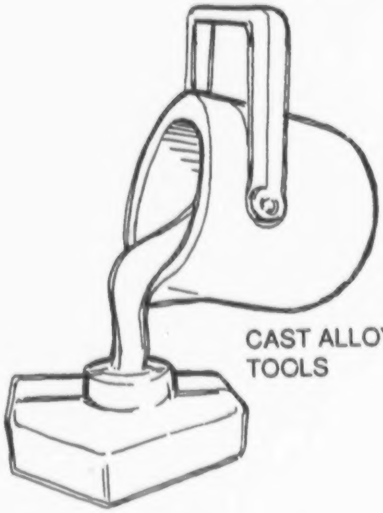


HIGH SPEED
STEEL LATHE
TOOL BLANKS

High-speed steel tools contain an alloy such as tungsten, molybdenum or cobalt.

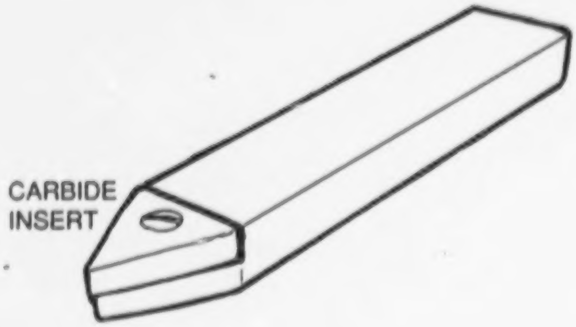


3. **Cast alloys**--available as inserts or brazed-on tips. They contain no iron. Main elements in these tools include tungsten, cobalt, chromium, nickel and carbon. You can operate cast alloy tools at speeds 50 percent higher than high-speed steel.



270

4. **Cemented or insert carbide tools**--an extremely hard cutting tool material composed mainly of tungsten and cobalt. These tools are widely used in factory machines. Carbide is available in a brazed-on tip or as a replaceable insert.



5. **Coated carbides**--carbide is sometimes coated with a thin layer of titanium nitride or silicon nitride. The tips may be gold colored. The coating helps the tool resist ware.

6. **Ceramic tools**--made by blending powders of ceramics and metals then heating to fuse the material

together. While harder than carbide, ceramic tools have less strength and are generally used on for light, finishing cuts.

7. **Diamond tools**--the hardest cutting tool material available. Diamond tools have low shock resistance and great strength. They are best used for cutting ceramics or other hard, abrasive materials. You can operate diamond tools at very high speeds.

DIAMOND
HARDEST
CUTTING TOOL
MATERIAL

F. WORKPIECE MATERIAL

The material of the workpiece affects which cutting tools you choose. The tool must be harder than the workpiece. You must also consider workpiece abrasiveness (its ability to wear away the tool). Certain metals and especially ceramics can wear away a cutting tool quickly.

ALUMINUM
COPPER
STEEL
STEEL

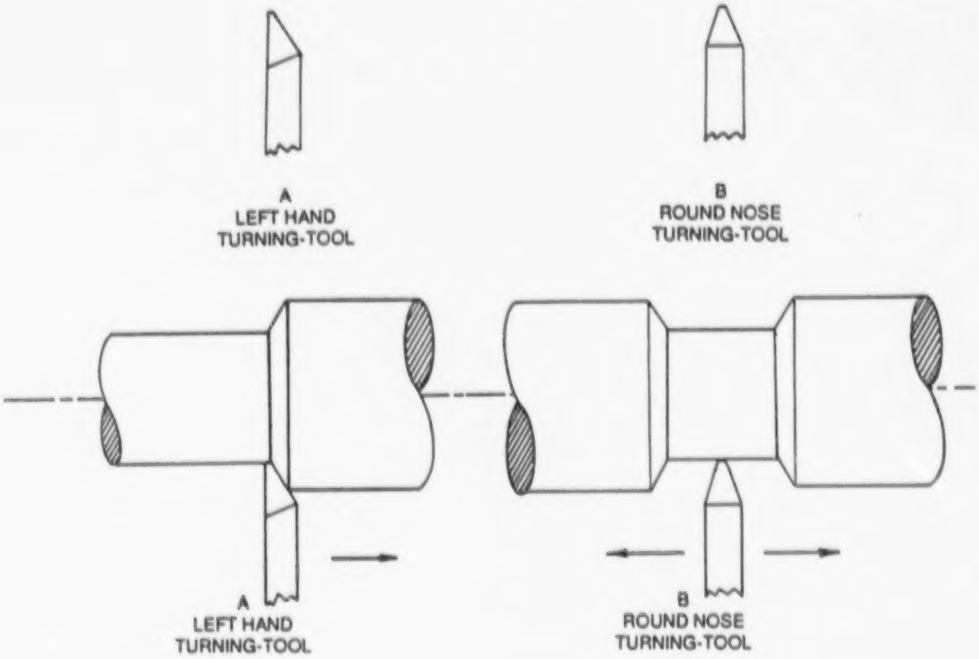
The list of materials below is in order from softest to hardest. As you move down the list, a harder, more expensive tool will be required for machining.

1. Soft plastics
2. Aluminum alloys
3. Zinc alloys
4. Copper alloys
5. Mild steel
6. Plain carbon tool steels
7. Alloy tool steels
8. Stainless steels
9. Ceramics

G. TYPES OF CUTTING TOOLS

Choosing the right tool for each job will make your work easier and better.

When turning on a lathe special tool forms are used to produce desired shapes and contours in the workpiece.


A
LEFT HAND
TURNING-TOOL
B
ROUND NOSE
TURNING-TOOL
A
LEFT HAND
TURNING-TOOL
B
ROUND NOSE
TURNING-TOOL

DO SKILL ACTIVITY 1

TOOL VARIABLES FOR MACHINING

ACTIVITY SHEET 1

From a selection of cutting tools provided by your instructor, identify the type of material in each.

From a selection of workpiece materials provided by your instructor, identify the type of material.

TOOL VARIABLES FOR MACHINING

UNIT TEST

Part 1. Match the terms below with their definitions.

A.	Red hardness	C.	Climb cutting
B.	Up milling	D.	Gibs

___ 1. A milling cutting action in which the cutter turns in the same direction as the workpiece is moving.

___ 2. A milling cutting action in which the cutter turns in the opposite direction from the motion.

___ 3. Thin pieces of metal between mating parts that are adjusted to take up normal wear on machine.

___ 4. A characteristic of a cutting tool (especially high-speed steel) that permits it to be operated at temperatures that turn the tool a dull red color.

Part 2. Match the cutting fluid types below with heir uses.

A.	Straight cutting oils	C.	Synthetic
B.	Water soluble		

___ 5. Used especially for cooling. Keeps the tools, machine and workpiece from rusting.

___ 6. Used for tapping, reaming and machining at high forces.

___ 7. Used to cool and lubricate in turning, grinding and milling.

TOOL VARIABLES FOR MACHINING

PERFORMANCE CHECKLIST

Student________________________________has successfully performed the following tasks:

	YES	NO	COMMENTS
1. Identified and selected tools based upon workpiece material and machine conditions.			
2. Matched cutting fluid types to their uses.			

Satisfactory - Should move on ☐

Repeat this unit ☐

Student's Signature/Date

Instructor's Signature/Date

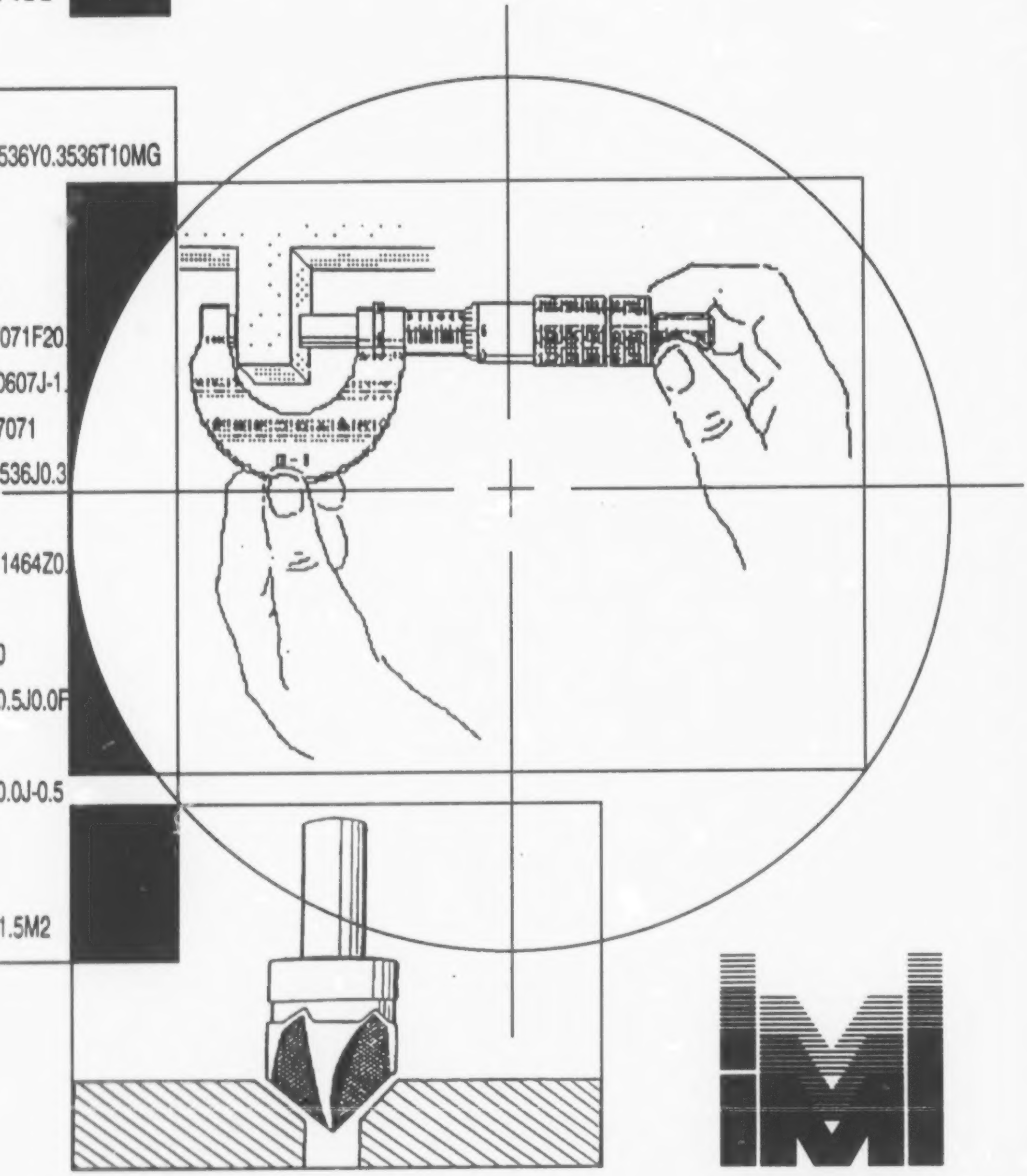
Machine Shop

90-9200-S
SUPPLEMENTARY UNITS

GRINDING FOR THE LATHE TOOLBIT

UNIT 9

```
%
N1691G00G75X0.3536Y0.3536T10MG
N2X0.0Y0.0Z0.0
N3Z0.05
N4G01Z-0.05F10.0
N5G01X0.7071Y0.7071F20
N6G03Y-2.1214I-1.0607J-1
N7G01X-0.7071Y0.7071
N8G02Y0.7072I-0.3536J0.3
N9G00
N10G00X1.6464Y1.1464Z0
N11Z0.05
N12G01Z-0.05F10.0
N13G03X-0.5Y0.5I-0.5J0.0F
N14G01X-3.0
N15G03X-0.5Y-0.5I0.0J-0.5
N20G01Y3.0
N21G00
N22G90G00X-2:0y-1.5M2
```



GRINDING THE LATHE TOOLBIT

INTRODUCTION

It is important for machinists to be able to grind their own cutting tools for the lathe from a toolbit blank. In this unit you will learn how to "off-hand grind" (grind by hand) a standard turning tool for lathework.

UNIT OBJECTIVE

After completing this unit, you should be able to grind a right-hand turning tool for use in the lathe.

SPECIFIC OBJECTIVES

After completing this unit, you should be able to:

1. Distinguish between left- and right-hand turning tools.
2. Inspect and adjust a pedestal grinder to assure proper operation.
3. Grind a right-hand turning tool with proper angles.

Put a check beside the statement that is true for you.

☐ I can do these tasks and want to go through the Performance Checklist.

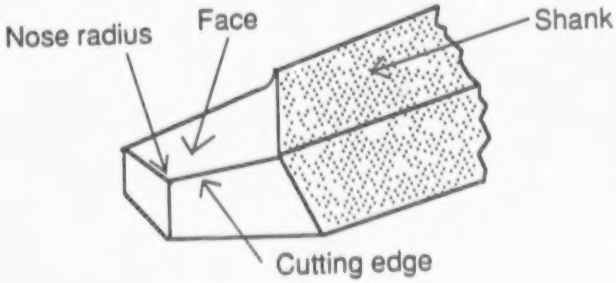
SEE YOUR INSTRUCTOR

☐ I want to study the information in this unit before I do the Performance Checklist.

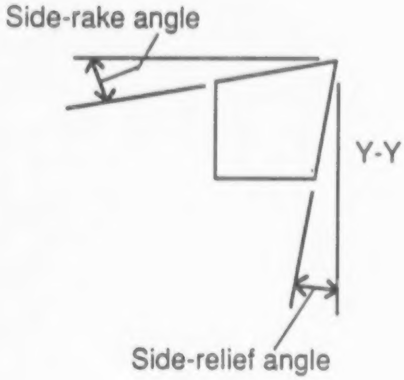
TURN PAGE AND BEGIN

A. TERMS

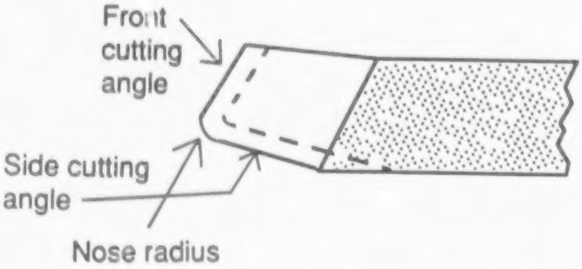
1. **Nose radius** - rounded portion on the end of a lathe toolbit.
2. **Face** - the top of the lathe toolbit.


Nose radius
Face
Shank
Cutting edge

3. **Side rake angle** - angle on the face of the tool that goes from side to side and guides the chip across the face of the tool.
4. **Side cutting angle** - angle of the side cutting edge.

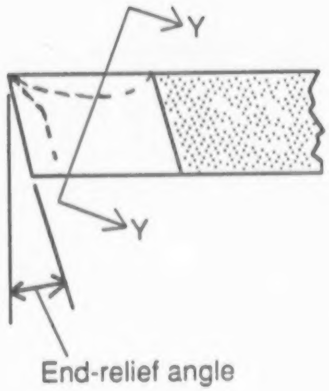

Side-rake angle
Y-Y
Side-relief angle

5. **Side cutting edge** - the sharp edge on the side of the toolbit.
6. **Front cutting angle** - the sharp edge on the front of the lathe toolbit.
7. **Back rake angle** - angle on


Front cutting angle
Side cutting angle
Nose radius

the face of the tool that goes from front to back and guides the chip across the face of the tool.

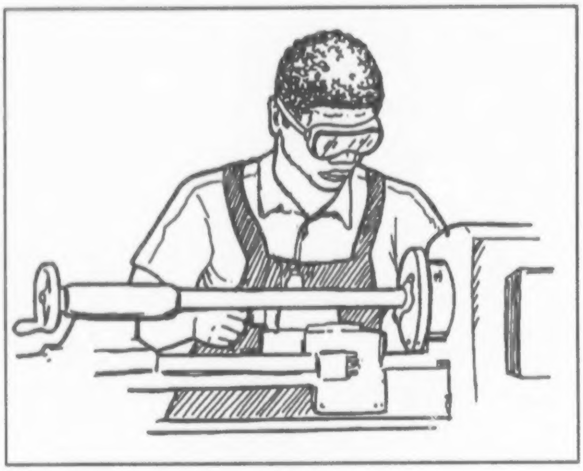
8. **End relief angle** - cut-away portion below on the tool end that forms the front cutting edge.



9. **Side relief angle** - cut-away portion on the side of the tool that forms the side cutting edge angle.

10. **Shank** - the body of the toolbit that is held in the toolholder.

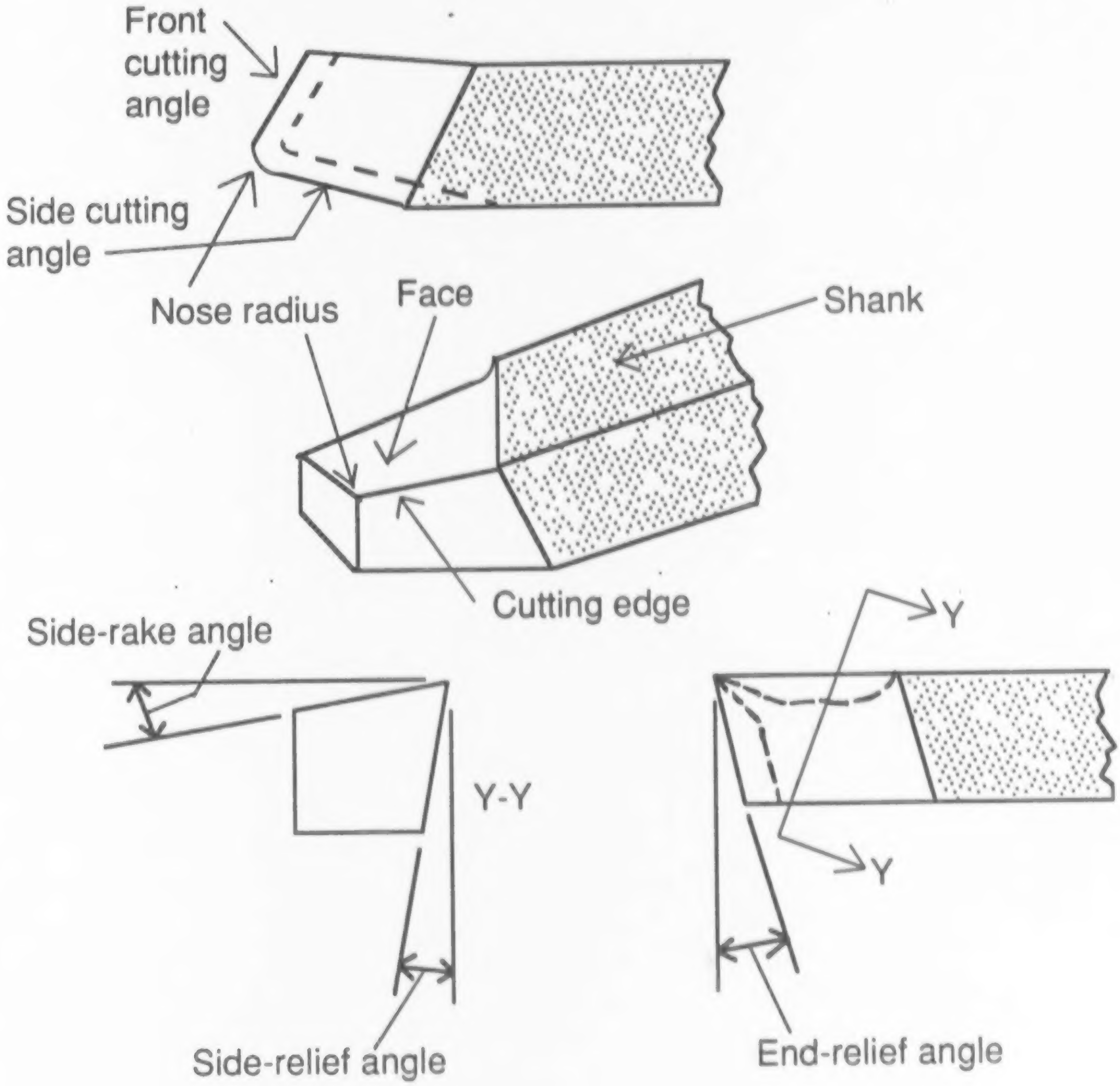
B. THE LATHE TOOLBIT



As a machinist, you must be able to grind a lathe toolbit. Only a properly ground toolbit will cut efficiently and produce a good finish.

You grind a toolbit "off-hand" by holding it in your hands and grinding the correct angles using a pedestal grinder.

C. PARTS OF THE LATHE TOOLBIT

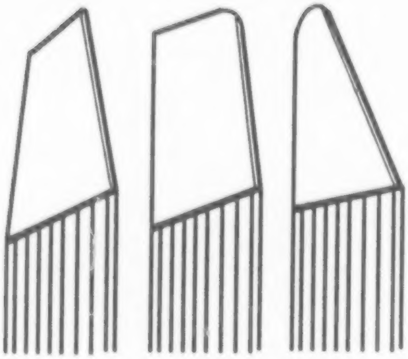

Front cutting angle
Side cutting angle
Nose radius
Face
Shank
Cutting edge
Side-rake angle
Y-Y
Y
Y
Side-relief angle
End-relief angle

D. SUGGESTED TOOLBIT ANGLES FOR CUTTING VARIOUS MATERIALS

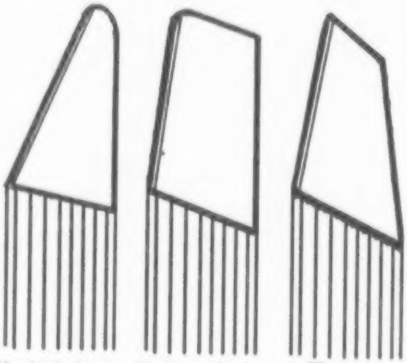
Suggested Cutting Angles for High-Speed Steel Tool Bits

MATERIAL	FRONT RELIEF ANGLE (°)	SIDE RELIEF ANGLE (°)	BACK RAKE ANGLE (°)	SIDE RAKE ANGLE (°)
High-Speed Steels (annealed)	10	6-8	8	12
Tool Steels (annealed) — alloy	10	6-8	8	12
Tool Steels (annealed) — high carbon	10	6-8	8	12
Stainless Steels (chip breaker a must)	10	6-8	8	12
SAE Steels: 1020	10-12	6-8	15	14
1045	10-12	6-8	12	14
1095	10	6-8	8	12
1320 (Mn)	10	6-8	15	15
2317 (Ni)	10	6-8	12	14
3120 (Ni - Cr)	10	6-8	12	14
4340 (Mo)	10	6-8	8	12
Aluminum	15-20	12-15	15-30	15-35
Bakelite — pressure-molded plastics	12	8	0	0
Brass — free cutting	10-12	8-10	0	0
— red and yellow	10-12	8-10	0	0
Bronze — cast, commercial	10-12	8	0-5	0-5
— free cutting	10-12	8	0	0
— hard	10	8	0	0
— phosphor	10	8-10	0	0
Cast Iron — gray	5-10	6-10	5-8	10-12
Celluloid — cast plastics	10-15	8-10	0-5	0
Copper	10-15	10-12	16	20
Fiber	10-15	10-12	0	0
Formica Gear Material	10-15	10	16	10
Monel Metal	10	6-8	8	14
Nickel	10	6-8	8	14
Nickel Silvers	10	6-8	0	0
Rubber — hard	10-15	10-15	0-3	0-7

E. LEFT-HAND AND RIGHT-HAND TOOLBITS


Facing
Roughing
Finishing

LEFT-CUT TOOLS


Finishing
Roughing
Facing

RIGHT-CUT TOOLS

DO SKILL SHEET 1

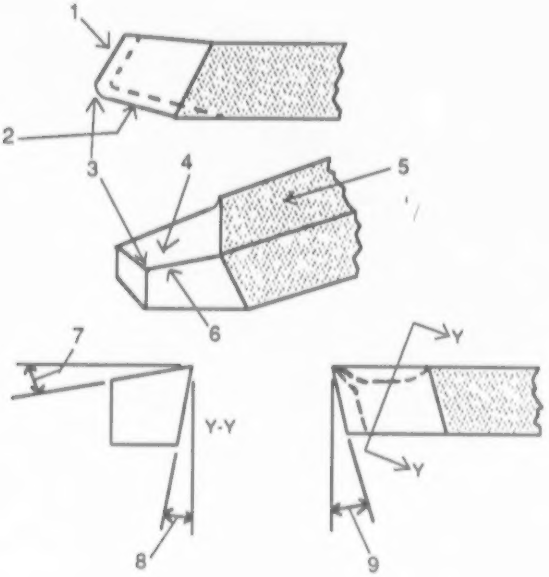
GRINDING THE LATHE TOOLBIT

SKILL SHEET 1

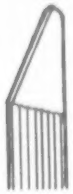
Match the parts of the lathe toolbit with the illustration below.

A. Front cutting angle
B. Shank
C. Nose radius
D. Side-rake angle
E. End-relief angle
F. Side cutting angle
G. Face
H. Side-relief angle
I. Cutting edge

___ 1.
___ 2.
___ 3.
___ 4.
___ 5.
___ 6.
___ 7.
___ 8.
___ 9.



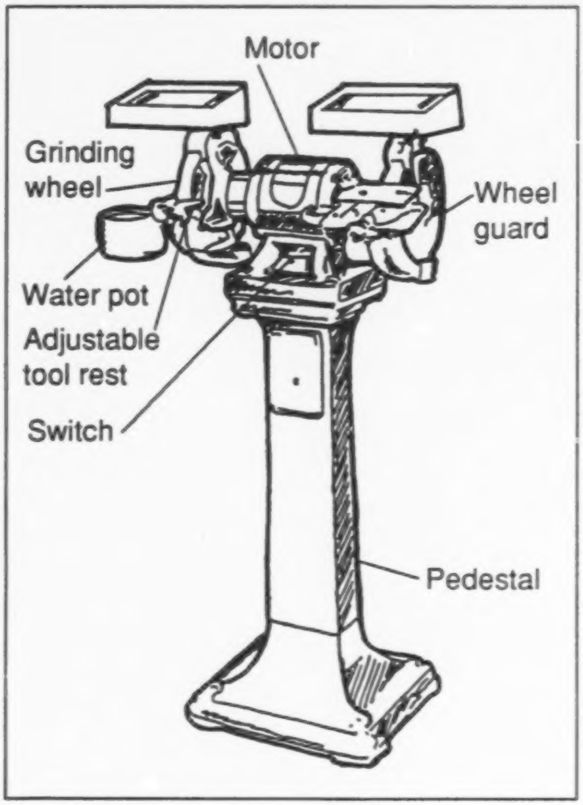
10. Is this tool a right-hand or left-hand tool?



F. THE PEDESTAL GRINDER

You use a pedestal or bench grinder to grind lathe toolbits. You will need to think about several things when you use a pedestal grinder:

1. How to use the grinder safely
2. How to adjust the grinder for safe operation
3. Wheel material
4. Toolbit material

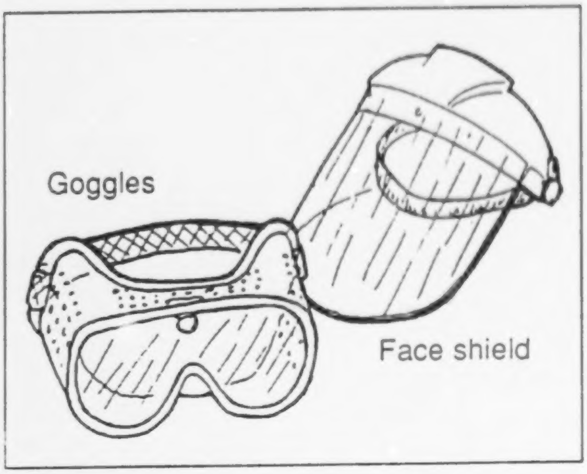

Motor
Grinding wheel
Wheel guard
Water pot
Adjustable tool rest
Switch
Pedestal

Note: Be sure to review the basic operation of the pedestal grinder. Don't use the grinder without your instructor's permission.

1. Pedestal grinder safety

Remember, always wear safety

goggles or a face shield when you are grinding. Also, be careful to keep your fingers away from the grinding wheel. Never grind on the side of the wheel. Wear short sleeves and remove all jewelry, rings and watches. If you have long hair, tie it back.

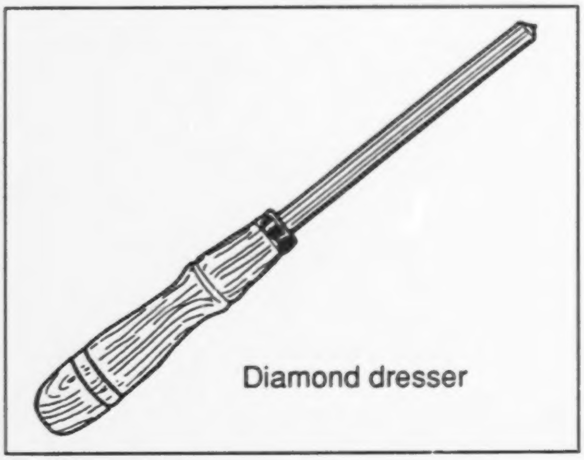


Note: Before you begin, have your instructor review your grinding skills to be sure you operate the grinder safely.

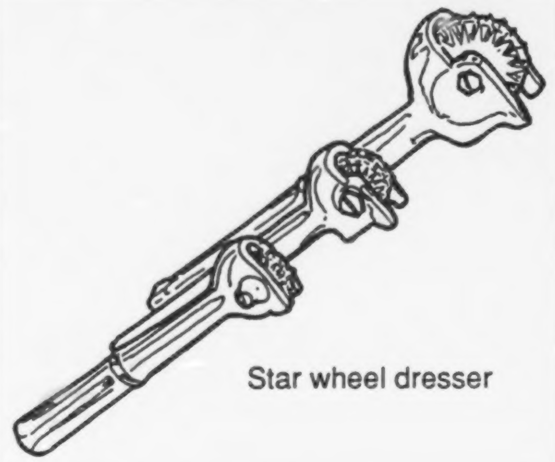
2. **Adjusting the grinder**

Adjusting the grinder takes two steps:

a. First, "dress" the wheel with a star or diamond dresser so that the grinding face is flat and square.



b. Second, adjust the toolrest so that it is no more than 1/16-inch away from the face of the grinding wheel.

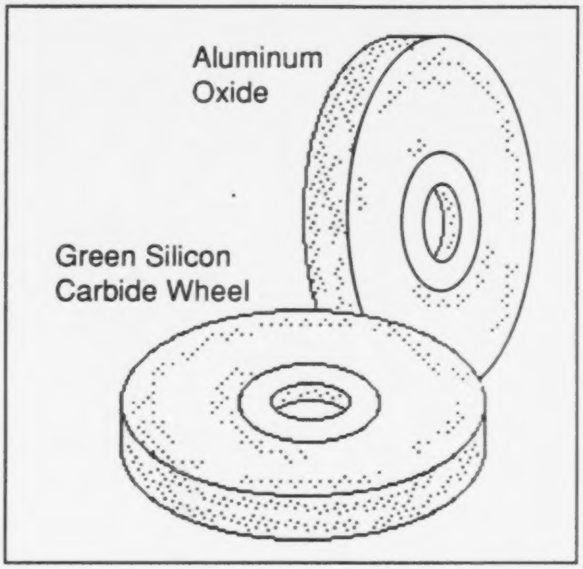

Star wheel dresser

CAUTION: Make certain power is off and wheel is not turning.

Also, fill the water pot because you will need it to keep the toolbit cool while you grind. If a water pot isn't available, keep a coffee can or other container of water close at hand. Keep the toolbit cool to the touch by dipping it in the water.

3. Grinding wheel material

It is important to use the right grinding wheel material with each toolbit. The most common toolbit materials are high-speed steel and carbide.


Aluminum Oxide
Green Silicon Carbide Wheel

For high-speed steel, use an aluminum oxide wheel (usually a gray color) of medium coarseness and hardness. For carbide, use a green wheel.

Note: Review grinder safety before handling any wheel.

DO SKILL SHEET 2

DO ACTIVITY SHEET 1

GRINDING THE LATHE TOOLBIT

SKILL SHEET 2

Choose the best response to answer each question.

___ 1. What type of wheel material should you use to grind high-speed steel?

A. aluminum oxide C. diamond

B. carbide

___ 2. What is the proper distance from the toolrest to the face of the grinding wheel?

A. 1/4" C. 1/16"

B. 1/8"

___ 3. Why should you have water available when you are grinding a lathe toolbit?

A. To mop the floor C. To drink

C. To keep the toolbit cool

GRINDING THE LATHE TOOLBIT

ACTIVITY SHEET 1

Your instructor will watch you adjust the toolrest to the proper distance from the grinding wheel.

G. PROCEDURE FOR GRINDING A RIGHT-HAND TURNING TOOL

While completing this unit, you will grind a standard right-hand turning tool with the following angles:

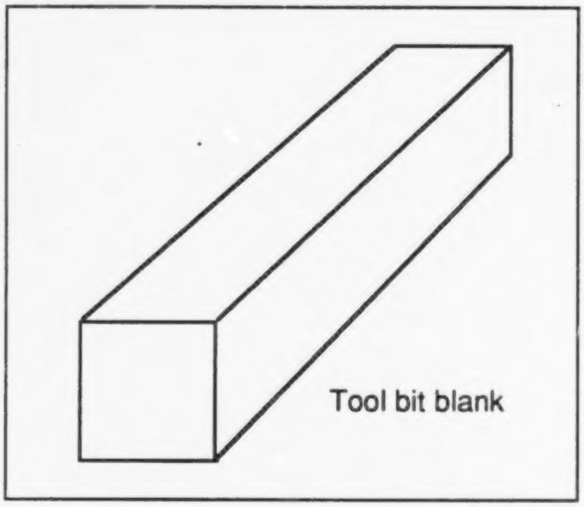
side rake angle	5 degrees
side cutting angle	8 degrees
side relief angle	10 degrees
front cutting angle	8 degrees
back rake angle	10 degrees
end relief angle	10 degrees

Grind angles in the following order:

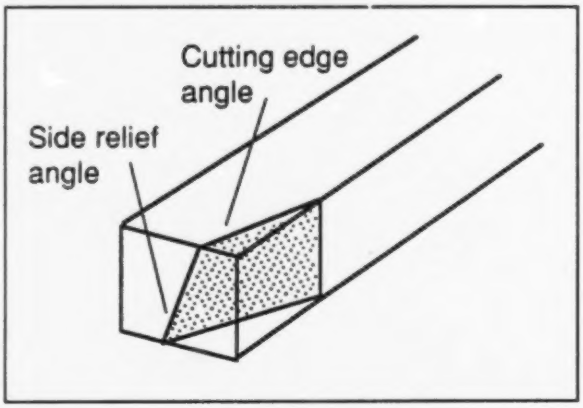
1. side cutting angle
2. side relief angle
3. front cutting angle
4. end relief angle
5. back rake angle
6. side rake angle

To grind a right-hand turning tool:

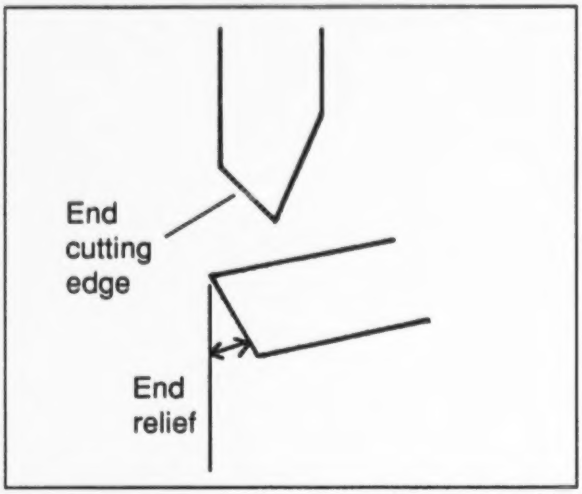
1. Obtain a 1/4- or 3/8-inch thick piece of keystock that is three to four inches square. You will use keystock for practice because it is much less expensive than a real toolbit blank. Hold the blank firmly while supporting your hands on the toolrest.


Tool bit blank

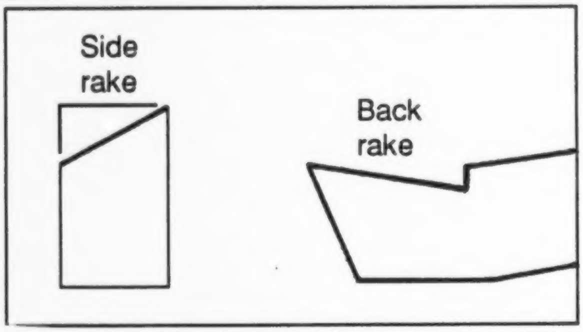
2. Turn the toolbit blank and grind the cutting edge angle. At the same time, turn the tool on its side about 10 degrees and grind the side relief angle.


Cutting edge angle
Side relief angle

3. Grind the end cutting edge. This is a combination of the front cutting angle and the end relief angle.


End cutting edge
End relief

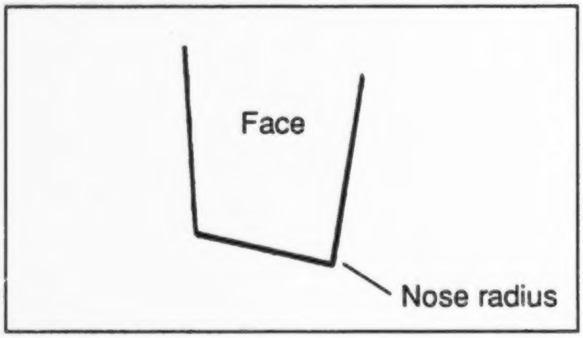
4. Hold the tool bit on its side and grind the side rake and back rake angles into the face (top) of the tool.


Side rake
Back rake

5. Grind nose radius between 1/32 and 1/16 inch.

6. Check angles with protractor.

7. Check radius with radius gage.


Face
Nose radius

DO SKILL SHEET 3

DO ACTIVITY SHEET 2

GRINDING THE LATHE TOOLBIT

SKILL SHEET 3

Write the proper angle for a standard turning tool.

1. Side rake angle __________
2. Side cutting angle __________
3. Side relief angle __________
4. Front cutting angle __________
5. Back rake angle __________
6. End relief angle __________

GRINDING THE LATHE TOOLBIT

ACTIVITY SHEET 2

HAND GRIND A RIGHT-HAND TURNING TOOL

Tools and equipment:

Safety goggles or face shield

1/4- to 3/8-inch keystock, 3 inches long

Pedestal grinder with aluminum oxide wheel

Water pot or can filled with water

Protractor to check angles

Instructions: Grind a standard right-hand turning tool with the following angles:

side rake angle	5 degrees
side cutting angle	8 degrees
side relief angle	10 degrees
front cutting angle	8 degrees
back rake angle	10 degrees
end relief angle	10 degrees

Grind angles in the following order:

1. side cutting angle
2. side relief angle
3. front cutting angle
4. end relief angle
5. back rake angle
6. side rake angle

Procedure: Evaluation

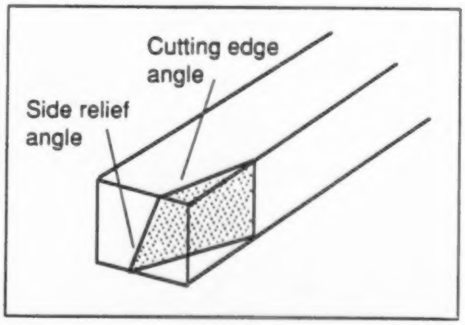
3 2 1 N

1. Hold the blank firmly while supporting your hands on the toolrest.

_ _ _ _

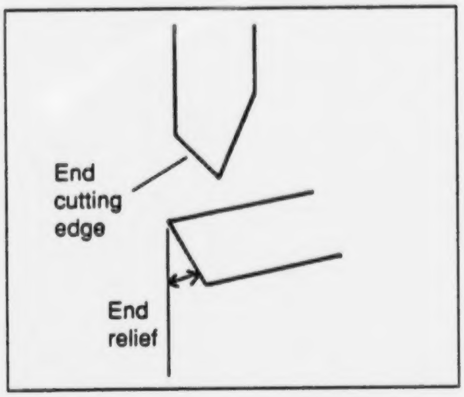
2. Turn the toolbit blank and grind the cutting edge angle. At the same time, turn the tool on its side about 10 degrees and grind the side relief angle.

_ _ _ _

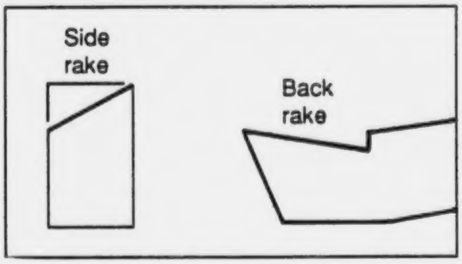


3. Grind the end cutting edge.

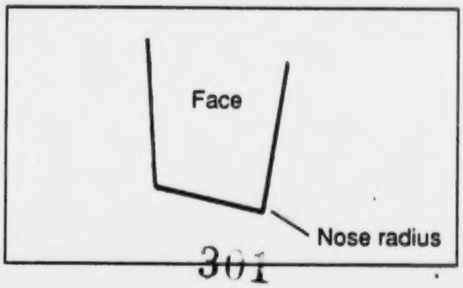
This is a combination of the front cutting angle and the end relief angle. _ _ _ _


End cutting edge
End relief

4. Hold the tool bit on its side and grind the side rake and back rake angles into the face (top) of the tool. _ _ _ _


Side rake
Back rake

5. Grind nose radius between 1/32 and 1/16 inch. _ _ _ _


Face
Nose radius

6. Check angles with protractor. – – – –

7. Check radius with radius gage. – – – –

8. Have instructor check your work. – – – –

GRINDING THE LATHE TOOLBIT

UNIT TEST

Part 1. Match the parts of the lathe toolbit with the illustration below.

- A. Front cutting angle
- B. Shank
- C. Nose radius
- D. Side-rake angle
- E. End-relief angle
- F. Side cutting angle
- G. Face
- H. Side-relief angle
- I. Cutting edge

___ 1.

___ 2.

___ 3.

___ 4.

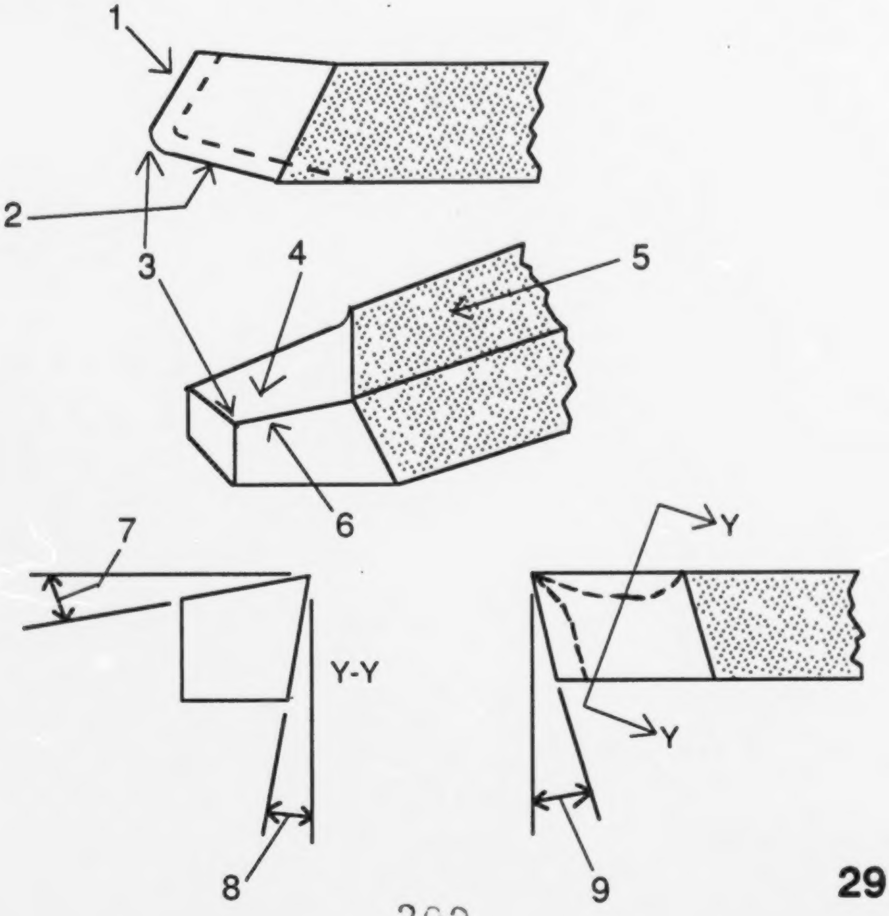
___ 5.

___ 6.

___ 7.

___ 8.

___ 9.



Part 2. Match the terms and definitions below.

A. Side cutting edge
B. Side rake angle
C. Nose radius
D. End relief angle
E. Side relief angle
F. Face
G. Shank
H. Side cutting angle
I. Front cutting angle
J. Back rake angle

___10. rounded portion on the end of a lathe toolbit.

___11. the top of the lathe toolbit.

___12. angle on the face of the tool that goes from side to side and guides the chip across the face of the tool.

___13. angle of the side cutting edge.

___14. the sharp edge on the side of the toolbit.

___15. the sharp edge on the front of the lathe toolbit.

___16. angle on the face of the tool that goes from front to back and guides the chip across the face of the tool.

___17. cut-away portion below on the tool end that forms the front cutting edge.

___18. cut-away portion on the side of the tool that forms the side cutting edge angle.

___19. the body of the toolbit that is held in the toolholder.

Part 3. Arrange the angles below in the order they would be ground.

A. end relief
B. side rake
C. side relief
D. side cutting
E. front cutting
F. back rake

___20.
___21.
___22.
___23.
___24.
___25.

GRINDING A LATHE TOOLBIT

PERFORMANCE CHECKLIST

Student____________________________has successfully performed the following tasks:

	YES	NO	COMMENTS
1. Distinguished between left- and right-handed turning tools.			
2. Inspected and adjusted a pedestal grinder to assure proper operation.			
3. Ground a right-hand turning tool with proper angles.			

Satisfactory - Should move on ☐

Repeat this unit ☐

____
Student's Signature/Date

____
Instructor's Signature/Date